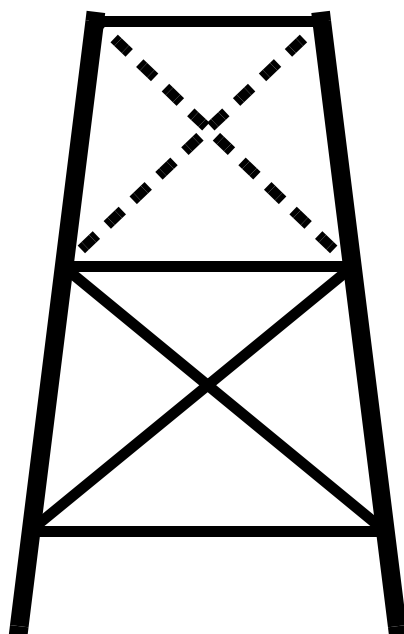




SESAM USER MANUAL

Preframe



Preprocessor for Generation of
Frame Structures

DET NORSKE VERITAS

SESAM

User Manual

Preframe

Preprocessor for Generation of
Frame Structures

September 10th, 2004

Valid from program version 6.9

Developed and marketed by
DET NORSKE VERITAS

DNV Software Report No.: 94-7096 / Revision 4, September 10th, 2004

Copyright © 2004 Det Norske Veritas

All rights reserved. No part of this book may be reproduced, in any form or by any means, without permission in writing from the publisher.

Published by:

Det Norske Veritas
Veritasveien 1
N-1322 Høvik
Norway

Telephone:	+47 67 57 99 00
Facsimile:	+47 67 57 72 72
E-mail, sales:	software.sesam@dnv.com
E-mail, support:	software.support@dnv.com
Website:	www.dnv.com

If any person suffers loss or damage which is proved to have been caused by any negligent act or omission of Det Norske Veritas, then Det Norske Veritas shall pay compensation to such person for his proved direct loss or damage. However, the compensation shall not exceed an amount equal to ten times the fee charged for the service in question, provided that the maximum compensation shall never exceed USD 2 millions. In this provision "Det Norske Veritas" shall mean the Foundation Det Norske Veritas as well as all its subsidiaries, directors, officers, employees, agents and any other acting on behalf of Det Norske Veritas.

Table of Contents

1	INTRODUCTION	1-1
1.1	Preframe — Preprocessor for Frame Modelling.....	1-1
1.2	Preframe in the SESAM System.....	1-2
1.3	How to read the Manual.....	1-4
1.4	Status List	1-4
2	FEATURES OF PREFRAME.....	2-1
2.1	Modelling Nodes and Elements.....	2-1
2.2	Modelling Properties.....	2-2
2.3	Conceptual data.....	2-3
2.4	Short Description of Commands.....	2-3
2.5	Transfer of the Model through the Input Interface File	2-6
2.5.1	Writing and Optimising the Input Interface File	2-7
2.6	Interaction with other SESAM Programs	2-8
3	USER'S GUIDE TO PREFRAME.....	3-1
3.1	Getting Started — the Graphical User Interface.....	3-2
3.2	Node and Element Modelling.....	3-7
3.2.1	Creating Nodes	3-7
3.2.2	Creating Elements	3-8
3.2.3	Degrees of Freedom (d.o.f.s) of a Node	3-10
3.2.4	Modelling Procedure for Jackets.....	3-10
3.2.5	Element Types.....	3-14
3.3	Properties	3-15
3.3.1	Eccentricity.....	3-15
3.3.2	Hinge	3-16
3.3.3	Local Coordinate System	3-17
3.3.4	Material	3-18

3.3.5	Cross Section	3-18
3.4	Align Elements.....	3-18
3.5	Members	3-19
3.6	Tubular Joint Modelling	3-20
3.6.1	Assign joint strengthening.....	3-20
3.6.2	Change joint geometry	3-21
3.6.3	Advanced gap calculations, command PROPERTY GAP	3-21
3.7	Hydrodynamic and Stability Data.....	3-26
3.8	Soil and Pile Modelling	3-26
3.9	Boundary Condition.....	3-28
3.10	Change Data.....	3-29
3.10.1	Change Nodal Coordinates.....	3-29
3.10.2	Change Load to Mass	3-29
3.11	Copy Data	3-30
3.12	Linear Dependency	3-30
3.13	Nodal Mass	3-31
3.14	Transformation.....	3-31
3.15	Load	3-32
3.16	Display Features.....	3-32
3.17	Printing Data	3-40
3.18	Writing input templates to soil stiffness and pile-soil analyses.....	3-40
3.19	Writing end-cut data for the jacket installation program	3-40
3.20	Writing and Reading Input Interface File	3-40
4	EXECUTION OF PREFRAME.....	4-1
4.1	Program Environment.....	4-1
4.1.1	Starting Preframe from Manager.....	4-1
4.1.2	Reading a Command Input File into Preframe.....	4-2
4.1.3	Starting Preframe as an Individual Program on Unix	4-3
4.1.4	Line-Mode Input of Commands and Arguments.....	4-3
4.1.5	Files used by Preframe	4-5
4.1.6	Creating Plots for Reports	4-6
4.1.7	Background Execution	4-6
4.1.8	Command Line Arguments	4-7
4.2	Program Requirements.....	4-9
4.2.1	Execution Time	4-9
4.2.2	Storage Space	4-9

4.3	Program Limitations	4-9
5	COMMAND DESCRIPTION	5-1
5.1	The Node and Element Select Features	5-2
5.2	Detailed Description of Commands.....	5-4
	ADD-DISPLAY	5-6
	ALIGN	5-8
	ASSIGN	5-9
	ASSIGN CAN.....	5-10
	ASSIGN CONE	5-12
	ASSIGN HYDRODYNAMIC.....	5-14
	ASSIGN PILE-DATA	5-16
	ASSIGN SEGMENT	5-18
	ASSIGN SOIL-DATA.....	5-19
	ASSIGN STABILITY	5-21
	ASSIGN STUB.....	5-23
	BOUNDARY	5-25
	CHANGE.....	5-27
	CHANGE CAN	5-29
	CHANGE CONE.....	5-31
	CHANGE JOINT.....	5-32
	CHANGE LINEAR-DEPENDENCY	5-33
	CHANGE LOAD.....	5-35
	CHANGE LOAD load-case TO-MASSSES	5-36
	CHANGE NODE.....	5-38
	CHANGE NODE ROTATE.....	5-39
	CHANGE NODE TRANSLATE	5-40
	CHANGE STUB.....	5-41
	COPY	5-43
	CREATE.....	5-45
	CREATE MEMBER.....	5-46
	CREATE MEMBER-FROM-ELEMENT	5-47
	DEFINE	5-48
	DEFINE GENSOD-DATA.....	5-49
	DEFINE SET	5-51
	DEFINE SOIL	5-53
	DELETE	5-54
	DELETE INITIAL-CONDITION	5-56
	DELETE LOAD	5-57
	DELETE PILE-CONCEPT	5-59
	DELETE SOIL	5-60
	DISPLAY.....	5-61
	ELEMENT.....	5-63
	ELEMENT GROUP: BEAM, TRUSS, NONSTRUCTURAL-BEAM, AXIAL-SPRING, AXIAL-DAMPER, GENERAL-SPRING and SHIM-ELEMENT.....	5-65

ELEMENT LINE: BEAM, TRUSS, NONSTRUCTURAL-BEAM, AXIAL-SPRING, AXIAL-DAMPER, GENERAL-SPRING and SHIM-ELEMENT	5-67
ELEMENT single: GENERAL-SPRING and SHIM-ELEMENT	5-69
ELEMENT single: SPRING-TO-GROUND and DAMPER-TO-GROUND	5-70
ELEMENT single: BEAM, TRUSS, NONSTRUCTURAL-BEAM, AXIAL-SPRING and AXIAL-DAMPER	5-72
EXIT	5-73
GENERATE	5-74
GENERATE eltyp JACKET	5-76
GENERATE eltyp K-BRACING	5-82
GENERATE eltyp LINE	5-84
GENERATE eltyp PILE	5-86
GENERATE eltyp PILE-FROM-SOIL	5-88
GENERATE eltyp T-BRACING	5-90
GENERATE eltyp X-BRACING	5-92
HELP	5-94
INITIAL-CONDITION	5-95
LABEL	5-96
LABEL CONCEPT-ATTRIBUTES	5-99
LABEL SOIL-DATA	5-101
LINEAR-DEPENDENCY	5-102
LINEAR-DEPENDENCY GENERAL-NODE-DEPENDENCY	5-104
LINEAR-DEPENDENCY TWO-NODE-DEPENDENCY	5-106
LOAD	5-107
LOAD load-case ELEMENT CONSTANT-TEMPERATURE-ACROSS-THICKNESS	5-109
LOAD load-case ELEMENT DISTRIBUTED	5-110
LOAD load-case ELEMENT LINE-LOAD	5-112
LOAD load-case ELEMENT POINT	5-113
LOAD load-case GRAVITY	5-114
LOAD load-case NODE FORCE	5-115
LOAD load-case NODE PRESCRIBED-ACCELERATION/-DISPLACEMENT	5-116
LOAD load-case ROTATION-OF-STRUCTURE	5-117
MASS-ON-NODE	5-118
NODE	5-119
NODE EXTRAPOLATION	5-120
NODE GROUP	5-122
NODE INTERSECTION	5-123
NODE LINE	5-124
NODE RELATIVE	5-126
NODE single	5-127
PLOT	5-128
PRINT	5-131
PRINT CONCEPT-ATTRIBUTES	5-132
PRINT DATA-CHECK	5-134
PRINT ECCENTRICITY	5-135
PRINT ELEMENT	5-136
PRINT HINGE	5-137

PRINT LOAD	5-138
PRINT LOAD OVERVIEW	5-139
PRINT LOAD SUM	5-140
PRINT LOAD load-case.....	5-141
PRINT LOCAL-COORDINATE	5-143
PRINT MASS-OF-ELEMENTS	5-145
PRINT MATERIAL	5-146
PRINT NODE.....	5-147
PRINT NODE BOUNDARY-CONDITIONS	5-148
PRINT NODE COORDINATES.....	5-150
PRINT NODE INITIAL-CONDITION.....	5-151
PRINT NODE LINEAR-DEPENDENCY	5-152
PRINT NODE MASS-ON-NODE	5-153
PRINT SECTION	5-154
PRINT SOIL	5-156
PRINT STATUS.....	5-158
PRINT STRUCTURE-CONCEPT	5-160
PRINT TRANSFORMATION	5-162
PROPERTY	5-163
PROPERTY CONNECT	5-164
PROPERTY ECCENTRICITY	5-165
PROPERTY GAP.....	5-166
PROPERTY HINGE.....	5-167
PROPERTY LOCAL-COORDINATE	5-169
PROPERTY MATERIAL	5-173
PROPERTY MATERIAL matno AXIAL-DAMPER / AXIAL-SPRING.....	5-174
PROPERTY MATERIAL matno DAMPER-TO-GROUND.....	5-175
PROPERTY MATERIAL matno GENERAL-SPRING	5-176
PROPERTY MATERIAL matno LINEAR-ELASTIC.....	5-177
PROPERTY MATERIAL matno SHIM-MEMBER.....	5-178
PROPERTY MATERIAL matno SPRING-TO-GROUND	5-179
PROPERTY SECTION	5-180
PROPERTY SECTION sctno BAR	5-182
PROPERTY SECTION sctno BOX	5-183
PROPERTY SECTION sctno CHANNEL.....	5-184
PROPERTY SECTION sctno DOUBLE-BOTTOM	5-186
PROPERTY SECTION sctno GENERAL	5-188
PROPERTY SECTION sctno I	5-190
PROPERTY SECTION sctno L	5-192
PROPERTY SECTION sctno PIPE	5-194
PROPERTY SECTION sctno UNSYM.-I.....	5-195
PROPERTY SOIL	5-197
RE-DISPLAY	5-198
READ.....	5-199
RENUMBER	5-200
ROTATE.....	5-201
SET	5-202

SET ALIGNMENT-AUTOMATIC	5-204
SET ASSIGN-OPTION	5-205
SET CAN-STUB-LENGTH-PARAMETER.....	5-206
SET COMMAND-INPUT-FILE	5-207
SET DEFAULT	5-208
SET ELEMENT-LOAD-DISTANCE-MODE	5-209
SET GRAPHICS.....	5-211
SET GRAPHICS AUTO.....	5-212
SET GRAPHICS CHARACTER-TYPE	5-213
SET GRAPHICS COLOUR	5-214
SET GRAPHICS DEVICE	5-215
SET GRAPHICS EYE-DIRECTION	5-216
SET GRAPHICS HIDDEN	5-217
SET GRAPHICS INPUT	5-218
SET GRAPHICS PLOT-FILE	5-219
SET GRAPHICS PRESENTATION.....	5-220
SET GRAPHICS SHRINK-FACTOR.....	5-221
SET GRAPHICS SIZE-SYMBOLS	5-222
SET JOURNALLING.....	5-224
SET MODEL-FILE	5-225
SET NUMBERING-AUTOMATIC.....	5-226
SET PLOT	5-228
SET PRINT	5-230
SET PRINT DESTINATION	5-231
SET PRINT FILE	5-232
SET PRINT FORMAT	5-233
SET PRINT PAGESIZE	5-234
SET PRINT TABLE NODE-BOUNDARY-TABLE.....	5-235
SET SOIL-PROFILE-X-Y	5-236
SET UNIT-VECTOR-TOLERANCE	5-237
SET WRITE-MODE.....	5-238
SPLIT	5-239
SPLIT ELEMENT-WISE	5-240
SPLIT NODE-WISE.....	5-242
TRANSFORMATION.....	5-243
WRITE	5-244
WRITE GENSOD-SPLICE-TEMPLATE.....	5-246
ZOOM.....	5-247
#.....	5-248

APPENDIX A TUTORIAL EXAMPLES..... A-1

A 1 Modelling a Module Frame	A-1
A 2 Modelling a Small 4 Legged Jacket with Soil and Piles.....	A-11
A 3 Result of CHANGE JOINT sel-nodes GAP-PLANEWISE	A-20

APPENDIX B THEORY B-1

B 1 Formulae for Sectional Parameters..... B-1

 B 1.1 Bar section B-2

 B 1.2 Box section B-3

 B 1.3 Channel section B-4

 B 1.4 Double-bottom section B-4

 B 1.5 I (or H) section B-5

 B 1.6 L section B-6

 B 1.7 Pipe section..... B-7

 B 1.8 Un-symmetrical I section B-8

B 2 Units..... B-9

 B 2.1 Example..... B-10

 B 2.2 Consistent Sets of Units B-11

REFERENCES REFERENCES-1

1 INTRODUCTION

1.1 Preframe — Preprocessor for Frame Modelling

Preframe is SESAM's preprocessors for modelling frame (beam), truss and cable models for hydrodynamic and/or structural analysis.

In addition to creating the structural model, i.e. data relevant for structural analysis (in Sestra), Preframe can be used for modelling certain hydrodynamic data relevant for hydrodynamic analysis (in Wajac) as well as certain joint and stability data relevant for code checking (in Framework).

Finally, Preframe may be used for modelling piles and soil relevant for structure-pile-soil interaction analysis (in Sestra and Splice).

Preframe is characterised by:

- Easy interactive input combined with graphical and printed feedback for model verification
- Extensive data generation features
- A data management system allowing arbitrarily large models

Input is interactively entered in Preframe. The user is guided by prompts for data and graphic functions are available for visualising model data. Data entered are logged on a command log (journal) file (commands not changing the model, for example a display command, are by default not logged). The log file can be used in a new Preframe session to regenerate the model. A standard text editor can be used to modify the log file for the purpose of creating a modified model. The log file is also a documentation and a backup of the modelling work.

Alternatively to interactive use Preframe can be run in batch (background) mode as explained in Chapter 4. For comprehensive modelling work you may find that editing an input file (which initially may have been a log file) is an efficient and complementary way of working to running Preframe in interactive mode.

Preframe creates either a complete model or a first level superelement constituting a part of the complete model. The difference between the two — as seen from Preframe — is that there are some supernodes (or super degrees of freedom) defined for the latter. The term 'superelement' is, nevertheless, also used for a

complete model made by Preframe, i.e. a first level superelement with no supernodes (or super degrees of freedom).

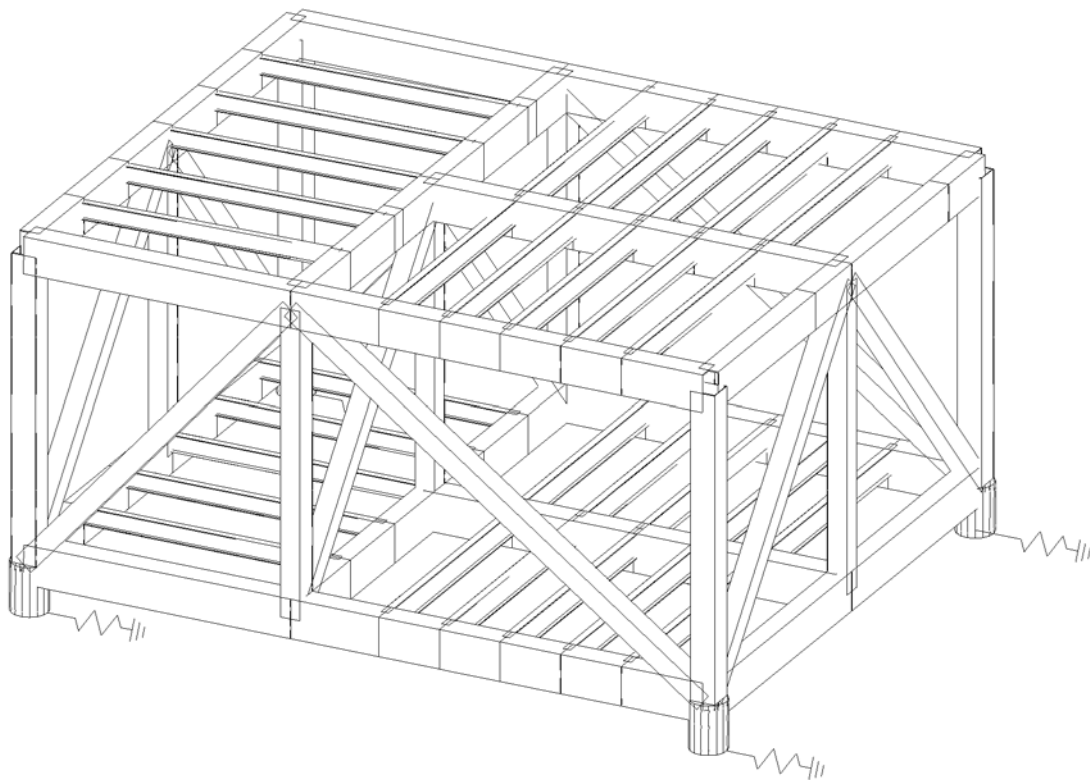


Figure 1.1 Example of a frame model created by Preframe

1.2 Preframe in the SESAM System

Additionally to Preframe, SESAM comprises a set of preprocessors that are dedicated to various modelling purposes. SESAM's preprocessors are:

Preframe	Modelling superelements consisting of beam, truss and cable elements
Prefem	Modelling superelements consisting of beam, membrane, shell and solid elements
Presel	Assembling superelements to form the complete model
Pretube	Modelling of tubular joints

In addition to these preprocessors SESAM is comprised of a set of hydrodynamic analysis programs, a set of structural analysis programs and a set of postprocessors. The SESAM system overview, an overview of all major SESAM programs and how they communicate, is shown in Figure 1.2.

The program Manager manages an analysis job including modelling, analysis and results processing by activating the proper programs and handling the files involved.

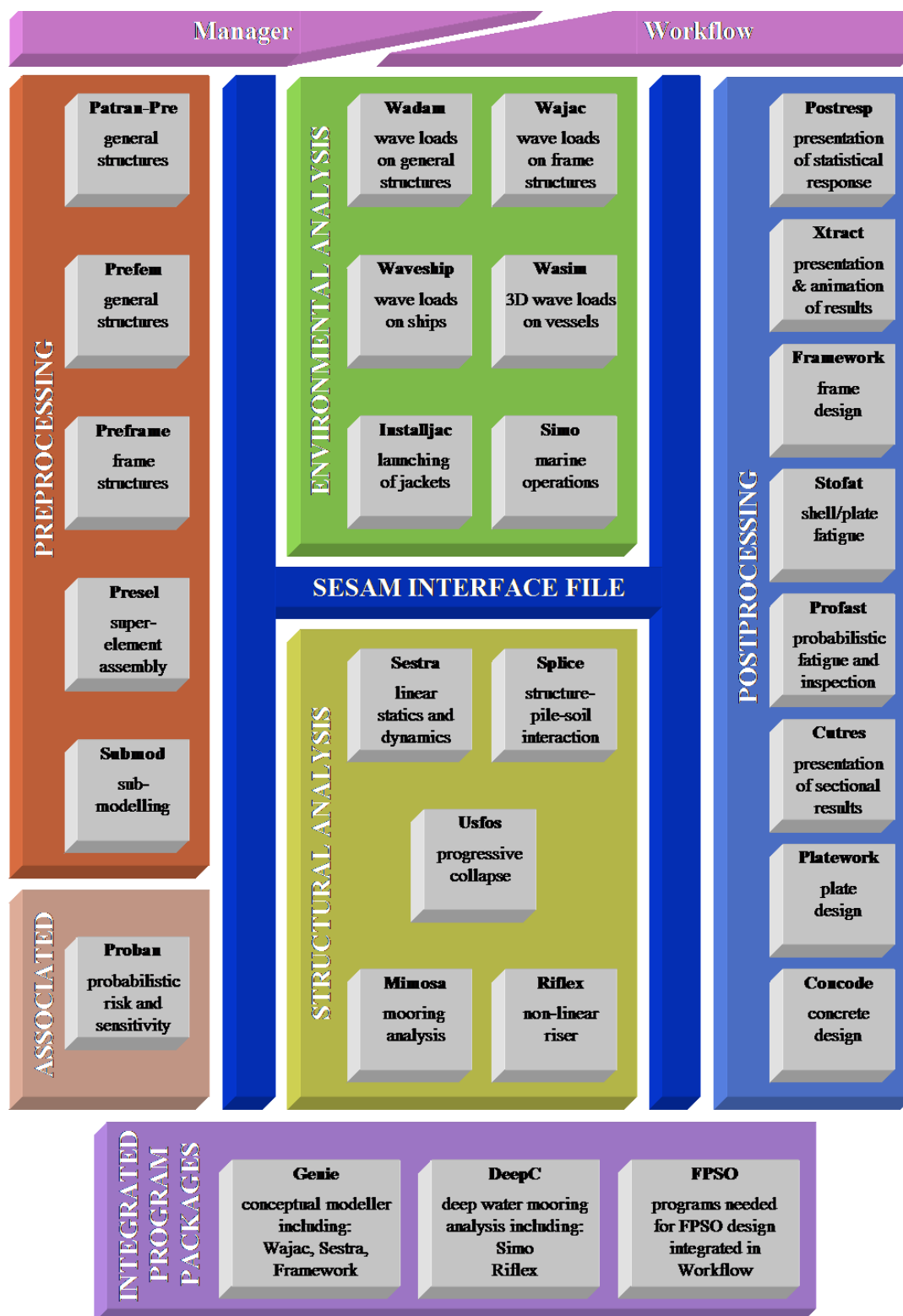


Figure 1.2 SESAM overview

1.3 How to read the Manual

Chapter 2 FEATURES OF PREFRAME contains an introductory description of the major features of Preframe.

Chapter 3 USER'S GUIDE TO PREFRAME explains how to create a complete model ready for analysis. All major features and several minor features are described. The chapter does not contain a full description of all program features, though; a complete understanding of all features of Preframe can only be obtained through training in use of the program while referring to Chapter 5.

Chapter 4 EXECUTION OF PREFRAME contains more special information not intended for the new user who will be using Manager to control his SESAM analysis. The chapter explains how to start Preframe outside Manager and operate it in line-mode (not using the graphical user interface). The files used by Preframe are also explained. Practical information is provided on how to operate Preframe and manipulate its files in various ways. Built-in and hardware dependent requirements and limitations are also described.

Chapter 5 COMMAND DESCRIPTION explains in detail all commands of Preframe. The commands and sub-commands are sorted alphabetically.

Appendix A TUTORIAL EXAMPLES contains a couple of examples of use.

Appendix B THEORY contains the formulae employed by the program for computing sectional parameters for the various types of beam cross section. Guidance in how to choose a consistent set of units for your analysis is also found here.

1.4 Status List

There exists for Preframe (as for all other SESAM programs) a Status List providing additional information. This may be:

- Reasons for update (new version)
- New features
- Errors found and corrected
- Etc.

To look up information in the most updated version of the Status List go to the support page of our website, click the SESAM Status Lists link and log into this service. Contact us for log-in information.

2 FEATURES OF PREFRAME

Preframe is a specialised interactive graphic program for modelling frame structures. Special features are available for efficient modelling of jacket type offshore structures.

2.1 Modelling Nodes and Elements

In Preframe the user creates nodes and elements using the `NODE` and `ELEMENT` commands, respectively. In addition to the simple creation of a single node with explicitly given coordinates it is possible to create a node relative to (offset from) an existing node, extrapolated from (or interpolated between) two existing nodes and at the intersection between two lines. Furthermore, several nodes may be created along a line and a group of nodes may be created. It is also possible to copy a set of existing nodes to a new position.

A model will always have a cartesian coordinate system to which all data refer. Some data, e.g. loads and boundary conditions, may be input in a transformed coordinate system but the data will always be converted to the model's cartesian coordinate system. The model's coordinate system is referred to as the *global* coordinate system.

Elements are either created one-by-one between existing nodes, along a line of existing nodes or a group of elements may be created. As for the nodes, existing elements may be copied to new positions.

Yet a way of creating nodes and elements is offered by the command `GENERATE`. This involves creating both nodes and elements in one operation. The command is highly efficient for jacket modelling in which the main structural components of *4, 6 and 8 legged jackets* may be created by a single command. Furthermore, a line of nodes and elements may be generated (rather than creating a line of nodes first and thereafter a line of elements) and various bracing (K, T and X) configurations may be inserted in a model.

The `GENERATE` command is also used for creating piles. The piles are modelled by ordinary nodes and beam elements. However, elements constituting the piles are recognised by being part of so-called pile concepts.

Nodes and elements are identified by user chosen numbers. The user chosen node numbers are termed external node numbers as opposed to the internal node numbers (see Section 2.5 for the difference between the two).

The following types of elements may be created:

- Two node beam element
- Two node truss element
- Two node axial spring and damper elements
- Single node spring and damper elements connecting the structure to the ground
- Two node general spring element
- Two node shim element (stiffness only in two translational directions)
- Two node non-structural beam element (contributes with mass and loads but has no stiffness)

2.2 Modelling Properties

Additionally to creating nodes and elements, the modelling consists of defining boundary conditions, cross sections, material types, local coordinate systems and a number of other model properties.

Loads acting in nodes or on elements can be defined. Also, acceleration fields like gravity and structure rotation (centripetal acceleration) that generate inertia loads may be defined.

The main commands for property modelling are:

- BOUNDARY for specifying fixations and other boundary conditions
- PROPERTY for defining:
 - Cross sections
 - Material types
 - Local coordinate systems
 - Eccentricities or rigid ends
 - Hinged connection between beam elements
 - Soil types (materials) for piled structures
- DEFINE for defining:
 - Sets
 - Soil profile, soil parameters and certain input data for the program Gensod (a part of Splice)
- CREATE for creating members out of beam elements
- ASSIGN for assigning:
 - Tubular joint strengthening data (cans, stubs and cones)
 - Hydrodynamic data (hydrodynamic coefficients and flooding information)
 - Stability data (buckling length and buckling factors)

- Segments to members (splitting into more elements while assembling in members)
- Pile data to pile concepts
- Soil stiffness data to the soil profile
- ALIGN for defining alignments of elements/members
- LOAD for defining loads

2.3 Conceptual data

Member concepts (segmented members) have been introduced to be able to hold information about several elements on a straight line between two nodes (structural joints). Advanced input commands exist to be able to quickly define can, stub and conical member segments in the model.

The member concept is also used to hold non-geometric information, i.e. hydrodynamic properties and stability parameters.

The member definition contains the following information:

- Element numbers between the two end nodes and in which order
- Information regarding elements representing can, stub and conical member segments
- Reference to hydrodynamic properties (C_d , C_m , flooding) assigned to the member
- Reference to stability parameters (buckling length, effective length factor) assigned to the member

2.4 Short Description of Commands

A short description of each main command of Preframe is given below.

ADD-DISPLAY	adds loads to the model display. The command also illustrates a soil profile by use of 'elements and nodes'.
ALIGN	specifies that two or more elements shall remain aligned, i.e. if an end node of a line of elements is moved then all nodes on the line is moved as well to maintain the alignment.
ASSIGN	assigns tubular joint strengthening data (cans, stubs and cones), hydrodynamic data, stability data, segments to members as well as pile and soil data.
BOUNDARY	defines boundary conditions for the nodes. Each d.o.f. of a node may individually be specified as FREE, FIXED, PRESCRIBED or SUPER. All d.o.f.s are by default FREE.
CHANGE	changes already defined data. Nodes, elements, loads, etc. may be changed. The command is also used to introduce eccentrici-

ties (offsets) to account for required gaps in tubular joints and to update can and stub lengths after joint modifications.

COPY

copies lines, planes or sets of nodes and elements.

CREATE

creates members out of beam elements.

DEFINE

defines sets of nodes and elements (such sets should not be confused with the SET command which sets various control parameters). Sets defined are written to the Input Interface File (see Section 2.5). Element sets may be referred to in Wajac. Both node and elements sets are transferred through the Sestra analysis program to the Results Interface File thus enabling the postprocessors to retrieve the sets. The command is also used to define soil profile, soil parameters and certain input data for the program Gensod (a part of Splice).

DISPLAY

displays the model. The nodes and elements may be displayed alone or combined. Member and pile concepts may be selected for display. Node and element numbers, boundary condition codes, etc. may be annotated using the LABEL command. Loads may be added by the ADD-DISPLAY command. In addition to the node and element displays, a 3-D view of a single joint may be displayed as well as its footprint (the chord-brace intersections on a developed chord).

ELEMENT

creates elements. The element type is chosen together with the nodes to which the element is connected.

GENERATE

creates both nodes and elements. In addition to generating a more or less complete jacket, a line of nodes and elements may be generated (rather than creating a line of nodes followed by a line of elements) and various bracing configurations may be inserted in a model. The command also creates piles.

HELP

provides information on the command syntax and how to get technical support. The command also launches the Status program providing access to the Status List of Preframe.

INITIAL-CONDITION

defines initial condition (at $t=0$) of nodes for forced response analysis by time integration.

LABEL

annotates node symbols, node numbers, etc. to the displayed model.

LINEAR-DEPENDENCY

makes one or more d.o.f.s of one node linearly dependent on one or more d.o.f.s of one or more independent nodes.

LOAD

defines loads for the model. The load cases are numbered successively from 1 and up. An arbitrary number of loads may be defined for the same node or element in the same load case.

MASS-ON-NODE	defines additional masses in nodes.
NODE	creates nodes with coordinates.
PLOT	generates a plot file of the last display or of the complete model. The plot file may be printed or imported in a word processor. In a MS Windows environment the plot may also be directed to an on-line printer.
PRINT	prints tables over model data. The tables may be directed to the screen or to a file by the SET PRINT command.
PROPERTY	defines and connects properties to elements. Such properties are cross section, materials, local coordinate systems, eccentricities (offsets or rigid ends), hinges and soil types.
RE-DISPLAY	displays the view as produced by the last DISPLAY command including possible load display (ADD-DISPLAY) and excluding annotations (LABEL).
READ	reads an Input Interface File containing a previously established model (first level superelement).
RENUMBER	re-numbers (changes the number of) previously defined nodes, elements, load cases, materials and cross sections.
ROTATE	rotates the display of the model.
SET	sets and defines various control parameters (this command should not be confused with sets of nodes and elements defined by the DEFINE SET command).
SPLIT	splits beam elements. Single elements may be split into any number of new elements. Elements connected to a common node may be split at given distances from the node, this feature is aimed at introducing cans and stubs for tubular joints.
TRANSFORMATION	defines a transformation from the model's (global) coordinate system. The transformation is used to describe transformed boundary conditions. It can also be used during input of for instance load data, the load data then refers to the transformed coordinate system rather than to the model's coordinate system.
WRITE	writes an Input Interface File containing the model (first level superelement). See Section 2.5 on this. The command is also used for producing templates for input to the programs Gensod and Splice.
ZOOM	increases or decreases the scale of the display.
#	reads commands from a command input file defined by the SET COMMAND-INPUT-FILE command.

DELETE	deletes data.
EXIT	exits from Preframe. The model and log files are saved and closed.

2.5 Transfer of the Model through the Input Interface File

As is the case for all SESAM preprocessors, the model created by Preframe is transferred to the hydrodynamic and/or structural analysis programs via the Input Interface File which forms a part of the SESAM Interface File system.

All information related to the member concepts are written to the SESAM Input Interface File, and the information is read by WAJAC and FRAMEWORK.

The Input Interface File, the T-file, is a sequential ASCII character file with 80 character long records. The straightforward definition of the file enables external programs to be connected to the SESAM system with comparative ease.

One interface file will be created for each superelement. The name of the file will be:

prefixT#.FEM

where:

- ‘prefix’ is an optional character string that may and may not include a directory specification, the string is common for all superelements in a superelement model.
- ‘T’ is a mandatory character identifying this as an Input Interface File, a T-file, as opposed to a Loads Interface File, L-file, which uses character L and a Results Interface File, R-file, which uses character R.
- ‘#’ is the superelement number, the identifier of the superelement.
- ‘FEM’ is a mandatory file extension.

Normally, the user may find it most convenient to leave the prefix void. This is also the default condition.

An example of a name of an Input Interface File is:

ABCT5.FEM

When using the superelement technique, all superelements belonging to the same model should have the same file prefix. If the above file — superelement 5 — is one of several files of a superelement model then all Input Interface Files should be named ABCT#.FEM, where # is the superelement number.

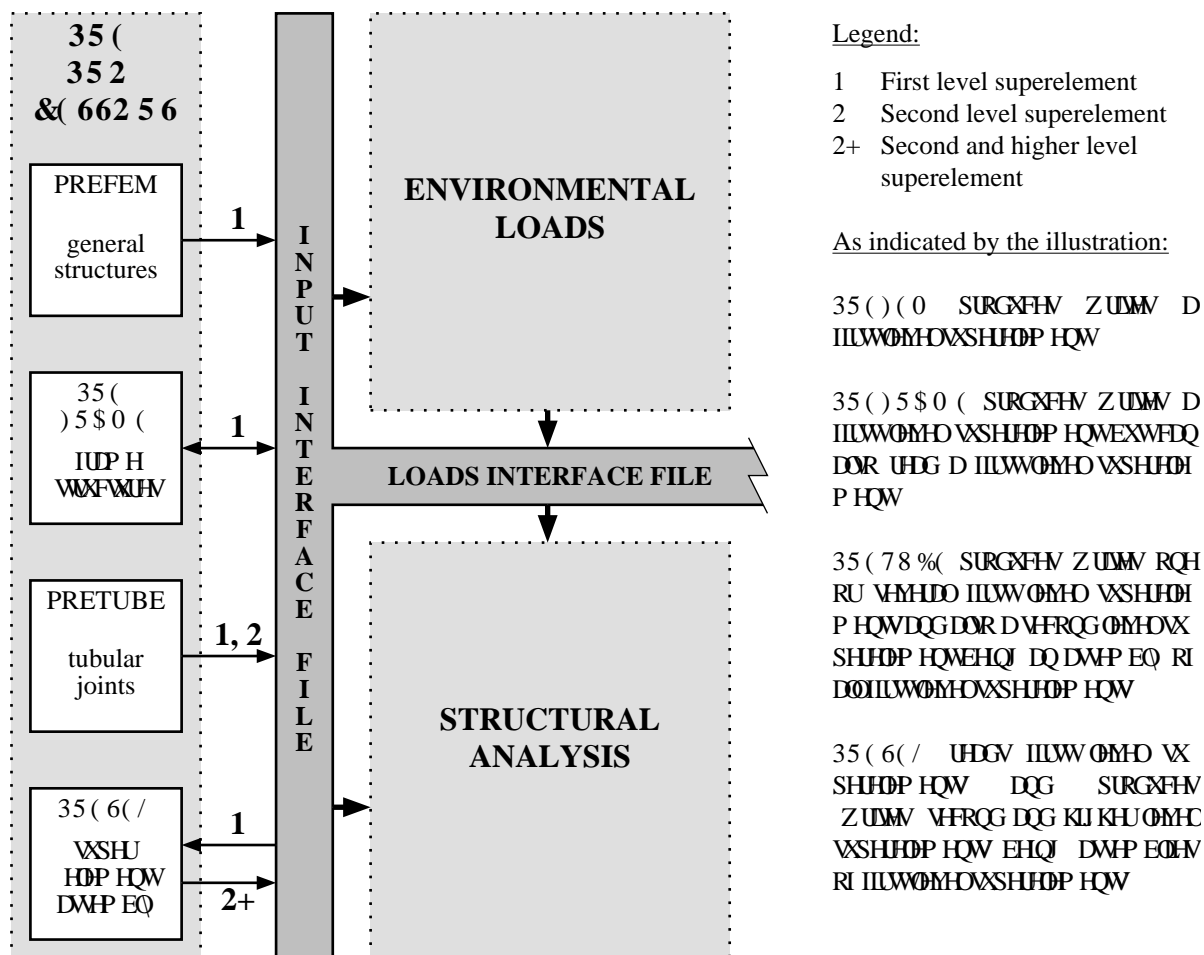


Figure 2.1 Interface between SESAM preprocessors and analysis programs

2.5.1 Writing and Optimising the Input Interface File

Whether or not to write the Input Interface File is normally controlled by Manager. If you want to produce the file you should check the appropriate box prior to starting Preframe. The Input Interface File is then automatically written when you exit Preframe using the command EXIT. This makes the command WRITE superfluous.

Note: If you on MS Windows close the Preframe window by the X in the upper right corner (or by the Close (Alt+F4) command of the window menu) then the Input Interface File will not be written even though you have requested this when starting Preframe. This feature may be used if you change your mind and decide not to write the file after having started Preframe.

If you run Preframe in a 'non-standard' way (not using the command Model | Frame Preframe in Manager) you need to either use a command line argument, see Section 4.1.8, or you need to use the command WRITE in order to produce the Input Interface File, the T-file.

When a static structural analysis using Sestra and its Multifront equation solver is to be performed the command will be:

```
WRITE #
```

However, if a dynamic analysis is to be performed in which case the Supermatrix equation solver of Sestra is employed the command will be:

```
WRITE BANDWIDTH-OPTIMIZATION #
```

This command changes (optimises) the internal node numbering (going from 1 to N, where N is the number of nodes in the model) in order to minimise the bandwidth of the stiffness matrix.

The auxiliary program Bpopt may also be used to optimise the bandwidth subsequently to writing the model to the Input Interface File, i.e. using the command WRITE # omitting the optional BANDWIDTH-OPTIMIZATION. Using Bpopt may be required if the model is too large for the optimisation facility inside Preframe (an in-core version).

Note: Unless Sestra's Multifront equation solver is used the optimisation (inside Preframe or outside using Bpopt) should *always* be performed or else the CPU time may be excessively large! Moreover, if the model created is a first level superelement which is to be coupled with other superelements then this optimisation should be performed prior to reading the model into Presel. **HOWEVER, DO NOT RUN BPOPT ON MODELS CONTAINING CONCEPTUAL INFORMATION (MEMBER AND ATTRIBUTES).**

2.6 Interaction with other SESAM Programs

All model characteristics that can be produced by Preframe are not necessarily accepted by a particular analysis program. Such consistency should be checked with the documentation of the relevant analysis program.

Code checks of beam elements may be done by the Framework postprocessor. There are some restrictions with respect to how the model is organised and which code checks are available. In particular, punching shear checks cannot be performed for joints that are supernodes. For further information refer to the Framework User Manual.

Preframe is able to read a model (first level superelement) from an Input Interface File. This is the case even if the model contains elements, loads, etc. that Preframe itself is unable to create. An example is a shell element model generated by Prefem. Such a model can be displayed, with hidden option if desired, and the data may be tabulated. It is not possible to change existing elements or loads, although nodal positions can be changed. Further, new nodes, elements (beams, trusses, cables, etc.) and loads can be added to the model.

3 USER'S GUIDE TO PREFRAME

This user's guide explains how to:

- Get started using the graphical user interface. See Section 3.1.
- Create nodes and elements. See Section 3.2.
- Define properties. See Section 3.3.
- Align elements. See Section 3.4.
- Create members. See Section 3.5.
- Model tubular joints (cans, stubs, cones and gaps). See Section 3.6.
- Assign hydrodynamic and stability data. See Section 3.7.
- Model soil and piles. See Section 3.8.
- Define boundary conditions. See Section 3.9.
- Change data. See Section 3.10.
- Copy data. See Section 3.11.
- Define linear dependencies. See Section 3.12.
- Define nodal masses. See Section 3.13.
- Define transformations to be used for defining boundary conditions, loads, etc. See Section 3.14.
- Define loads. See Section 3.15.
- Display and print data. See Section 3.16 and Section 3.17.

3.1 Getting Started — the Graphical User Interface

Preframe is started from the SESAM Manager by clicking **Model | Frame Preframe**.

See Section 4.1.3 for how to start Preframe outside Manager (Unix only).

The main part of the graphical user interface is the graphic-mode window. The appearance of this window is principally the same on PC (Windows) and Unix. On PC there are also a print window and a message window. Print requested by the user appears in the print window whereas various program messages appear in the message window. Figure 3.1 illustrates the three Preframe windows on a PC.

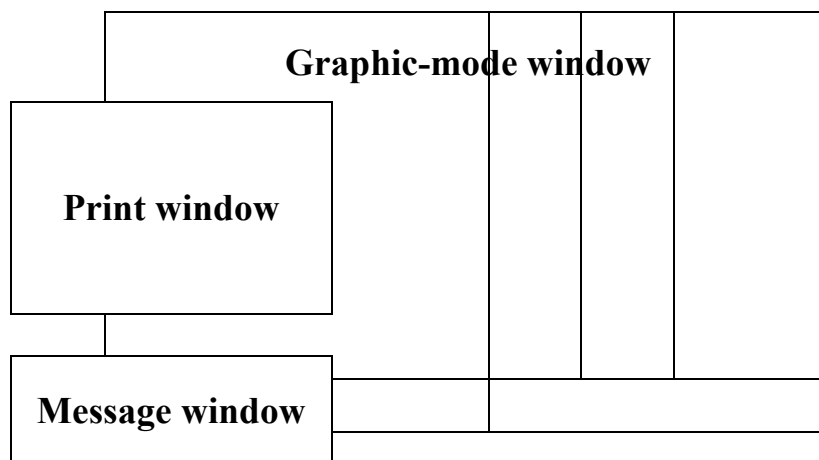


Figure 3.1 The Preframe windows on PC

On Unix there is only a line-mode window in addition to the graphic-mode window. I.e. the print and message windows are replaced by a line-mode window where print requested by the user as well as program messages appear. (The line-mode window is also where line-mode input is entered if you do not use the graphical user interface; see Section 4.1.4 on this.)

Preframe offers two modes of input and both are available in the graphic-mode window:

- Line-mode input, i.e. typing commands and data using the keyboard
- Graphic-mode input, i.e. selecting commands by clicking the left mouse button (LMB)

A sketch of the graphic-mode window is shown in Figure 3.2 together with explanations of the six different areas. How to use the areas is explained in more detail in the following.

You may at this stage decide to read about how to create nodes and elements. Go then to Section 3.2 and use the explanations of the areas of the graphic-mode window below for reference.

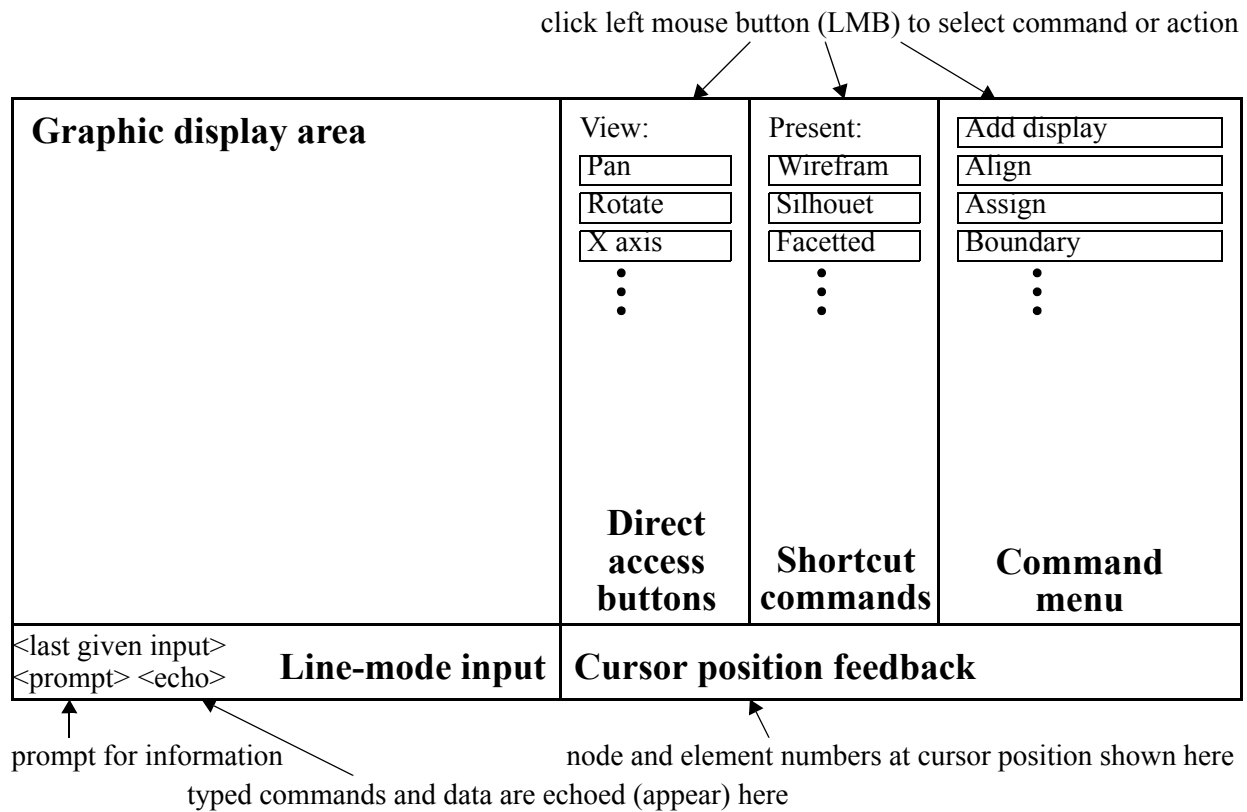


Figure 3.2 The graphic-mode window is composed of six different areas

The six different areas of the graphic-mode window are used as follows:

- **Graphic display area**

- The model is displayed here. The display is automatically updated when new nodes and elements are defined (unless this feature is on purpose switched off). See Section 3.16 for some display examples.
- Within several commands there is a need for selecting nodes or elements, e.g. when boundary conditions are defined and when loads are defined. Alternatively to keying in the nodes and elements as explained in Section 5.1 you may select nodes and elements graphically. There are three ways of doing this:
 - Clicking the left mouse button (LMB)
 - Dragging a rubber-band rectangle using the LMB
 - Polygon selection: Position the cursor and press the shift key to define the first polygon point. While keeping the shift key pressed repeatedly move the cursor and click the LMB to make a polygon. Release the shift key and click to define the last polygon point. A straight line between the first and last polygon points closes the polygon. If the LMB is pressed rather than clicked a rubberband line appears as an aid to determine the position of the polygon segment.

- The availability of graphical selection is subject to that node/element selection has been switched on by the Direct access buttons Node and Element. By default they are both switched on (depressed). See information on these buttons below.
- Note that if the Direct access button Info is depressed then nodes and elements *cannot* be selected by clicking. See information on the Info button below.

• Command menu

- The at any time allowable commands plus default values for numerical data are listed here as buttons.
- Commands and values are selected by clicking the left mouse button (LMB).
- Slanted text signifies default choices that are accepted by either:
 - Hitting the Return key
 - Clicking either of the Direct access buttons ‘;’ (semicolon) and ‘/ /’ (double slash). The former accepts all subsequent default values (see Section 4.1.4) while the latter accepts a single default value, i.e. the one shown in slanted font.

• Shortcut commands

These provide one-click access to commonly used compound commands. A Shortcut command is logged as its equivalent full standard commands. The Shortcut commands are sorted in six groups as follows:

— Present:

- **Wirefram** — Beams are displayed as lines (wireframe model)
- **Silhouet** — Beams are displayed as silhouettes
- **Facetted** — Beams are displayed in facet mode, i.e. as ‘solid’ objects

— Display:

- **Joint** — Display a selected joint in 3-D with intersection lines
- **Plane** — Display a selected plane of the model, i.e. all nodes and elements in a plane
- **Foot** — Display the foot print of a joint
- **All** — Display all nodes and elements of the model
- **Soil On** — Add display of the soil profile
- **Soil Off** — Remove display of the soil profile

— Generate:

- **Line** — Generate a line, i.e. create nodes and elements along a line between two existing nodes
- **Jacket** — Generate a whole jacket
- **K brace** — Generate a K-bracing
- **T brace** — Generate a T-bracing
- **X brace** — Generate an X-bracing
- **Pile** — Generate piles
- **Pile<soi** — Generate piles based on an existing soil model

— Element:

- **Line** — Create elements along a line of existing nodes

— Node:

- **Line** — Create nodes along a line between two existing nodes

— Label:

- **Elem Num** — Add element numbers to the display
- **Node Num** — Add node numbers to the display
- **Sect Num** — Add section numbers to the display
- **Matr Num** — Add material numbers to the display
- **Mem Nam** — Add member names to the display
- **Pile Nam** — Add pile names to the display
- **Soil Typ** — Add soil types to the display
- **Boundary** — Add boundary condition symbols to the display

- **Direct access buttons**

These buttons are accessible at any time. I.e. when you are in the middle of a command (by clicking a command or a Shortcut command or by typing a line-mode command) you may rotate and zoom to get a better view. The buttons ‘;’ and ‘//’ are logged with the default values they accept. The button ‘..’ is logged as is. The other buttons are not logged (see Section 4.1.5 on logging commands). The Direct access buttons are sorted in three groups as follows:

— View:

- **Pan** — allows panning (shifting) the display. Click the button, then press and hold the LMB within the Graphic display area and a bounding box of the displayed model appears. Move the mouse and release the LMB and the model will be displayed in its new position.
- **Rotate** — allows interactive rotation of the display. Click the button, then press and hold the LMB within the Graphic display area and a bounding box of the displayed model appears. Move the mouse up and down to rotate the model about a screen horizontal axis and move left and right to rotate about a screen vertical axis. A circular motion will rotate the model about an axis normal to the screen in the *opposite* direction of the circular motion. When the LMB is released the model is displayed in its new position.
- **X axis, Y axis and Z axis** — display the model as seen along the model’s X-, Y- and Z-axis, respectively.
- **Default** — switches back to the default viewing position (optionally set in Manager) and re-displays the model.
- **Zoom In** — zooms in by either clicking twice and diagonally or by pressing the LMB and dragging it to form a zoom area (rubber-band box).
- **Zoom Fr** — re-displays the model so that it fits within the display area.
- **Refresh** — refreshes the display with the last setting.

— Misc:

- **Learn** — offers making a new Shortcut command. Click the button and enter a maximum eight character string being the name of the new Shortcut command and hit Return. Now give any sequence of commands. Several complete commands may be given, the last of which may be incomplete (i.e. more data is required to make it complete). Clicking the learn button once more completes the process and the new Shortcut command appears as a new button.

- **Info** — offers quick information on nodes and elements. When clicked (depressed) the program enters into an info-mode involving that clicking nodes and elements provides information (coordinates, etc.) on the clicked items. The information appears in the print window (line-mode window on Unix). Note that when in info-mode nodes and elements cannot be selected by clicking; dragging rubberband still functions as selection though. Click the button once more (lift it) to leave info-mode. Combined with pushing the Shift key, this function will give you distance between nodes and angles between beam elements.
- **;** — button accepts all available default commands and parameters.
- **..** — button aborts the current command.
- **//** — button accepts a single default value, i.e. the one shown in slanted font.

— Select:

- **Node** — button (under heading Select) must be depressed (the default condition) to allow graphical selection of nodes. Also, the Cursor position feedback, see below, only works when the Node button is depressed.
- **Element** — button (under heading Select) must be depressed (the default condition) to allow graphical selection of elements. Also, the Cursor position feedback, see below, only works when the Element button is depressed.
- **Set** — button (under heading Select) is merely a consequence of GUI consistency with other SESAM preprocessors and has little relevance for Preframe.

• Line-mode input

- The upper line presents the last given input.
- The lower line includes the prompt for input and data entered in line-mode.
- On PC you may paste (Ctrl+V) text into the line-mode input area.

• Cursor position feedback

- The node and element numbers at or close to the cursor position are listed here. If more than one node/element is within the tolerance of the cursor position then all these nodes/elements will be listed.
- This Cursor position feedback only works when the Node/Element direct access buttons (under heading Select) are depressed. This may be utilised as follows: If you cannot tell which is which of node and element numbers because there are several numbers listed you may click (lift) the node (or element) button. Then only elements (or nodes) will be listed.

Note: The line-mode window (only on Unix) is inaccessible for entering data after having given the SET GRAPHIC INPUT ON command.

Note: While entering a command by the keyboard it is not possible to click buttons or commands until hitting the Return key or deleting all data typed. This involves that if you (inadvertently) have entered a ‘space character’ (which you may overlook as you cannot see it) clicking commands as well as selecting nodes and elements by clicking will not work. Use the backspace to delete the ‘space character(s)’.

Note: Graphical selection of nodes and elements does not work if the Info button is depressed. You will then instead get information on the nodes/elements. See the explanation of the Info button above.

3.2 Node and Element Modelling

3.2.1 Creating Nodes

Nodes are created by the NODE command and are identified by user chosen, maximum seven digit (integer) numbers. Alternative ways of creating nodes are shown below. Figure 3.3 illustrates the result of these examples.

Note: The command SET NUMBERING-AUTOMATIC ON allows for automatic generation of node numbers in which case the node numbers are omitted in the commands below.

In the examples below indents are used merely to ease the readability while x, y, z, xx, etc. represent coordinate values that may be given as integer or real values.

The data entries may be given one by one (i.e. hitting return between each entry) or with several entries separated by one or more blanks as shown below (a comma may also be used to separate entries). However, only one complete set of data, e.g. one node with coordinates, may be given on a single line. Double-dot (..) will exit the current command while preserving complete sets of data.

First create the two single nodes 101 and 901:

```
NODE 101 x y z
      901 xx yy zz
..
```

Now create a line of nodes (201 401 601 801) between the two existing nodes 101 and 901. Note that the *relative* distances between the nodes are given (i.e. not necessarily the absolute distances). The program prompts and default values (between slashes) are shown in bold:

```
NODE LINE 101 901
NO. OF DIVISIONS? /2/ 5
  4 NODE NUMBERS? 201 401 601 801
  5 SPACINGS? /EVEN/ 0.6 4 4 4 0.6
..
```

Create nodes 411 and 611 relative to the existing nodes 401 and 601:

```
NODE RELATIVE 401 dx dy dz 411
              601 dxx dyy dzz 611
..
```

Create node 506 at the intersection between the two lines 401-611 and 411-601:

```
NODE INTERSECTION 401 611 411 601 506
..
```

Create node 11 extrapolated along a line between nodes 201 and 101 until the line hits a given XY-plane, i.e. a given value of the Z-coordinate:

```
NODE EXTRAPOLATION 201 101 XY-PLANE-INTERSECT z-val 11
..
```

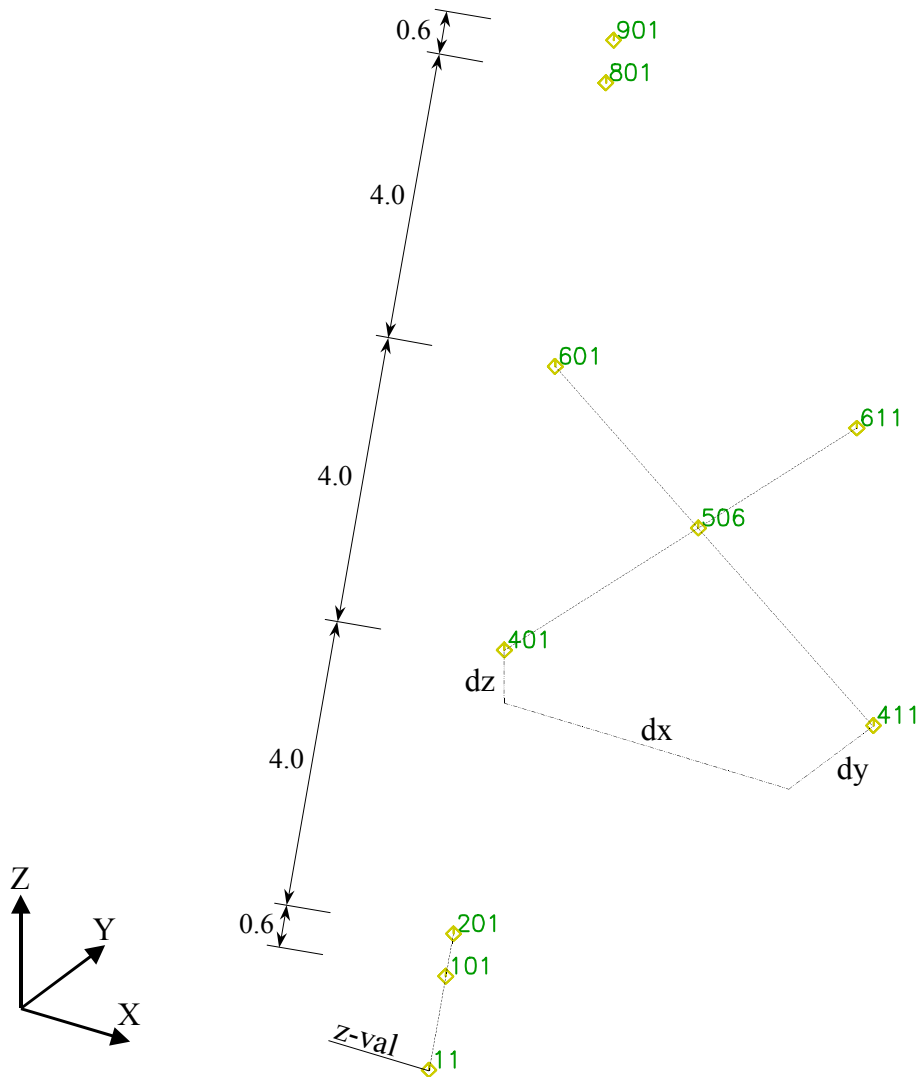


Figure 3.3 Nodes are created by the NODE command

3.2.2 Creating Elements

Elements are created by the ELEMENT command and are identified by user chosen, maximum seven digit (integer) numbers. Elements can only be created between pairs of nodes that previously have been created (or attached to a previously created single node in case of spring- and damper-to-ground elements). Alternative ways of creating elements are shown below. Figure 3.4 illustrates the result of these examples.

Note: The command SET NUMBERING-AUTOMATIC ON allows for automatic generation of element numbers in which case the element numbers are omitted in the commands below.

First create a single beam element (element 11):

```
ELEMENT BEAM 11 101 201
..
```

Then create a line of beam elements (elements 21, 41, 61, 81) along a line of nodes from node 201 to node 901:

```
ELEMENT BEAM LINE 201 901
21 41 61 81
..
```

Note: The local beam x-axis is defined as pointing from the first to the second node as given in the element definition command.

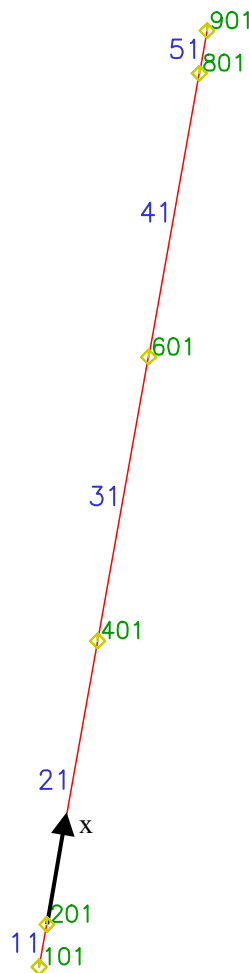


Figure 3.4 Elements are created by the ELEMENT command, local x-axis defined at the same time

3.2.3 Degrees of Freedom (d.o.f.s) of a Node

The number of degrees of freedom (d.o.f.s) of a node will always be either three (three translations) or six (three translations plus three rotations). The number of d.o.f.s is decided by the element with the highest number of d.o.f.s among the elements connected to the node.

When giving boundary conditions and loads you must give data corresponding to all six d.o.f.s irrespective of the actual number of d.o.f.s. That is, you must enter dummy values for non-existing d.o.f.s.

3.2.4 Modelling Procedure for Jackets

For modelling jackets the GENERATE JACKET command is recommended as the most efficient way of establishing the model. The command creates the major part of the main structural components of a jacket. The remaining modelling, typically refining the model by inserting bracings, conductor frames and so forth is easily done once the framework of the jacket has been established. The command has the following characteristics:

- Four, six and eight legged jackets may be created.
- All legs and the main bracings are created.
- The jacket may have any number of elevations (horizontal bracings).
- Conductors or only the conductor nodes (in case conductors will be a separate superelement) may be included in the model. (The conductor frame is not created and must be added afterwards.) A set named CONDUCT containing all conductor nodes and elements is automatically created to ease later reference to this part of the model.
- The jacket may be inclined (the centre of the top is positioned with an offset compared to the bottom).

The data to give within the command are:

- Main dimensions like length, width and Z-coordinate of the bottom and top of the jacket
- Elevations (Z-coordinates) of the horizontal bracings
- Type (presently limited to X-bracing) and position of additional bracing
- Vertical conductors, only their X- and Y-coordinates are given, each conductor will have a node at each elevation
- Section numbers (previously defined) assigned to the various elevations of legs, bracings and conductors
- Top offset of the jacket if the jacket is inclined, i.e. a horizontal offset of the top of the jacket compared to the bottom (the centre of the bottom of the jacket will always have zero X- and Y-coordinates)

When specifying bracing, note the following:

- A row number in longitudinal direction identifies the panels on both sides of the jacket.
- The transverse row identifies all transverse panels of the jacket, i.e. two panels for a four legged, three panels for a six legged and four panels for an eight legged jacket.

Below is an example of a command for generating an eight legged jacket. The cross sections 1, 2, 3 and 4 need to be created as these are referred to within the GENERATE command. d1, t1, d2, etc. are the diameters and wall thicknesses of the tubular sections. For each section two shear factors also need to be given, these are normally set to 1.0.

```
% First create cross sections referred to in the GENERATE command:
```

```
PROPERTY SECTION 1 PIPE d1 t1 1.0 1.0
                  2 PIPE d2 t2 1.0 1.0
                  3 PIPE d3 t3 1.0 1.0
                  4 PIPE d4 t4 1.0 1.0
```

```
..
```

```
GENERATE BEAM JACKET 8-LEGGED
```

```
% Give the main dimensions:
```

```
80.0 50.0 0.0 60.0 30.0 75.0 18.0
```

```
% Give elevations (Z-values) for horizontal bracings:
```

```
5.0 26.0 48.0 70.0 END
```

```
% Specify X-bracings in row 3, elevations 2 and 3:
```

```
BRACINGS X-BRACINGS 3 2 3 END
```

```
% And X-bracings in transverse row elevation 1:
```

```
TRANSVERSE-ROW 1 END END
```

```
% Define conductors in (X,Y)=(11,-4) and =(11,-2):
```

```
CONDUCTORS 11 -4 11 -2 END NONSTRUCTURAL-BEAM
```

```
% Assign section 1 to legs, all elevations:
```

```
SECTIONS LEGS ALL-ELEVATIONS 1 END
```

```
% Assign section 2 to horizontal bracings, all elevations:
```

```
HORIZONTAL-BRACINGS ALL-ELEVATIONS 2 END
```

```
% Assign section 3 to X-bracings, all elevations:
```

```
X-BRACINGS ALL-ELEVATIONS 3 END
```

```
% Assign section 4 to conductors:
```

```
CONDUCTORS 4
```

```
END END END
```

```
..
```

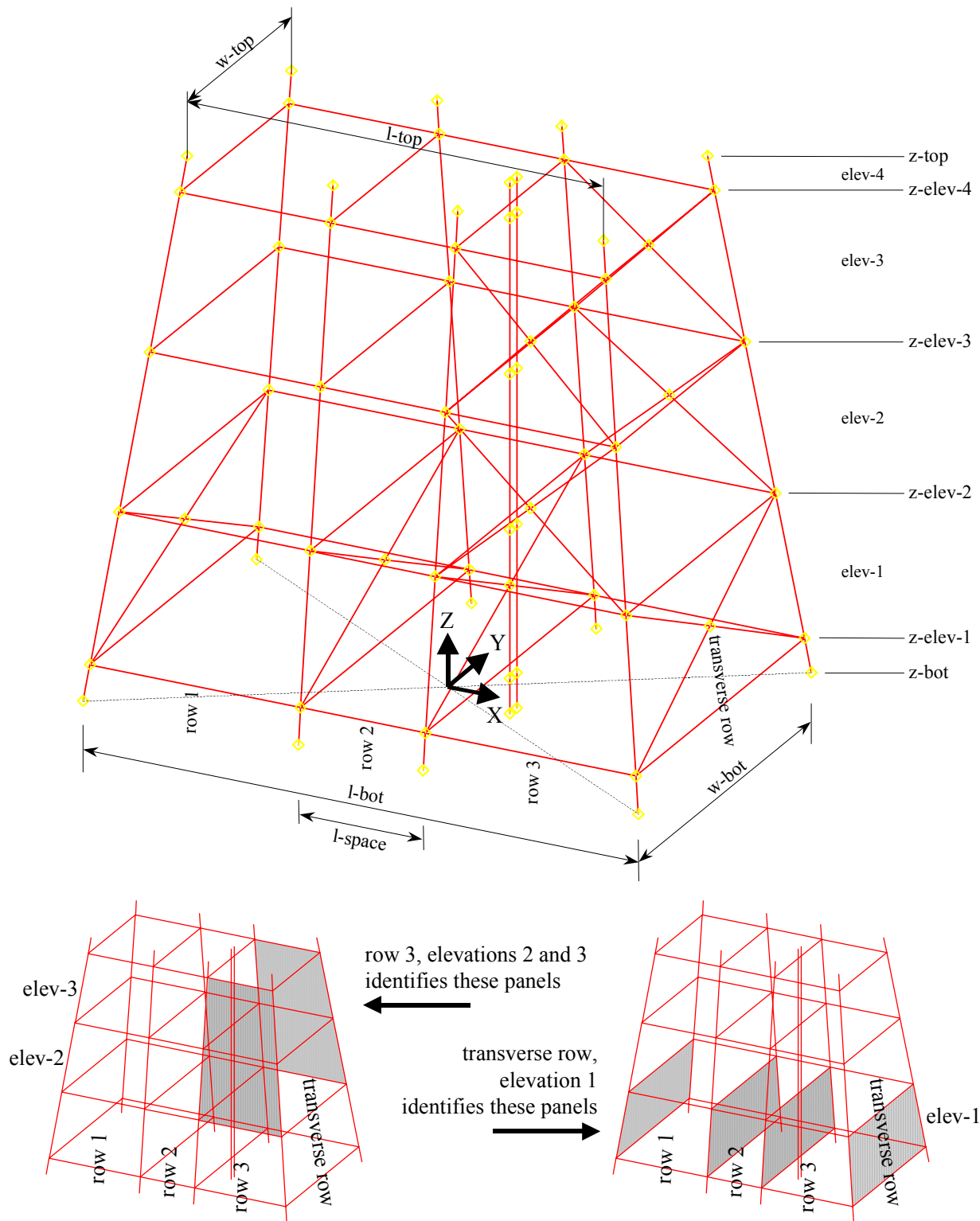


Figure 3.5 Jacket model created by GENERATE command

How to model conductors as a separate superelement

When creating conductors by the GENERATE BEAM JACKET command you may alternatively only create the nodes. Combined with the automatically defined set named CONDUCT this feature may be used to create the jacket and conductor superelements in a very efficient way. Do this as follows:

- 1 Generate the jacket superelement:
 - a Use the GENERATE BEAM JACKET command to create the jacket and for the conductors select only nodes to be created.
 - b Define the conductor nodes as supernodes by referring to the set CONDUCT within the BOUNDARY command.
- 2 Generate the conductor superelement (remember to change the superelement number):
 - a Generate a new jacket using the same input as above only replacing the NODES-ONLY option with BEAMS or NONSTRUCTURAL-BEAM as desired. Also, remember to assign section numbers to the conductors.
 - b Define a set containing all nodes except for the nodes contained in the set CONDUCT. This is done by the command
`DEFINE SET NOTCOND UNION NODE ALL SUBTRACT-BY NODE SET CONDUCT NO END`
 - c Delete all nodes and elements except for the conductors. This is done by deleting the set of nodes named NOTCOND (deleting nodes also deletes elements connected to the nodes):
`DELETE NODE SET NOTCOND NO`
 - d Define all nodes as supernodes (or all nodes belonging to the set CONDUCT).

These two superelements may then be assembled using Presel. As they are defined using the same origin there is no need for shifting or rotating them relatively to each other when assembling them.

How to model a conductor frame and use a set for copying it between elevations

The GENERATE BEAM JACKET command optionally creates the nodes and elements of the conductors. The conductor frame, however, is left for the user to model. You may find it convenient to model the conductor frame by the commands NODE RELATIVE / EXTRAPOLATION / INTERSECTION combined with the commands GENERATE BEAM LINE / T-BRACING and other commands for creating nodes and elements.

If the conductor frames of the various elevations are partly identical you may model the identical part only once and copy it to the other elevations as follows:

- 1 Model one of the conductor frames, e.g. the one at elevation 1. Model only the part that is identical with the other elevations.
- 2 Define a set (see the DEFINE SET command) containing the conductor frame elements that are to be copied. (The set need not contain nodes as these will be copied, if necessary, when elements are copied.)

- 3 Copy the set (see the COPY SET command). Nodes and elements coinciding with existing nodes and elements will not be copied. The copying is based on an increment in the node and element numbering; this implies that a numbering system is required to avoid conflict in numbers between the different elevations.
- 4 Complete the modelling for each elevation, i.e. model the parts that differ between the elevations and therefore cannot be copied.

How to model structural joint strengthening and conical transitions

Add structural joint strengthening sections (cans / stubs) by use of the command ASSIGN CAN and ASSIGN STUB.

Add / insert conical transitions to a member by use of the command ASSIGN CONE.

Update joints with brace eccentricities due to minimum gap by use of the command CHANGE JOINT <select-nodes> GAP-PLANEWISE.

Update can and stub lengths due to change in joint lay-out by use of the command CHANGE JOINT <select-nodes> CAN-STUB-LENGTH.

How to assign conceptual attributes

Use the ASSIGN STABILITY command to add member stability parameters (buckling length and effective length factor). The stability parameters will be read by FRAMEWORK.

Use the ASSIGN HYDRODYNAMIC command to add hydrodynamic properties (Cd, Cm, flooding) to the member concepts. The hydrodynamic properties will be read by WAJAC. The flooding parameter will also be read by FRAMEWORK.

3.2.5 Element Types

The element types that may be created are (see also Table 5.3 on page 5-66):

BEAM-(BEAS)	Two node beam element A linear elastic material must be defined and assigned (connected) to the element. The element also requires a cross section and a local coordinate system.
TRUSS-(TESS)	Two node truss element with no bending stiffness A linear elastic material must be defined and assigned (connected) to the element. The element also requires a cross section but only the area of the cross section is relevant. A local coordinate system is irrelevant.
NONSTRUCTURAL-BEAM-(BEAS/N)	Two node element not contributing with stiffness The element only requires a cross section if wave loads are to be computed (by Wajac). A material (density) is also required if the element shall contribute with mass to the structural analysis.

AXIAL-SPRING-(AXIS)	Two node axial spring element An axial spring material must be defined and assigned (connected) to the element.
AXIAL-DAMPER-(AXDA)	Two node axial damper element for dynamic analysis only An axial damper material must be defined and assigned (connected) to the element.
SPRING-TO-GROUND-(GSPR)	One node spring element A spring to ground material must be defined prior to creating the element as the material is referred to when creating it.
DAMPER-TO-GROUND-(GDAM)	One node damper element for dynamic analysis only A damper to ground material must be defined prior to creating the element as the material is referred to when creating it.
SHIM-ELEMENT-(GLSH)	Special two node spring element with equal stiffness in two translational directions, no stiffness in the third translational direction and no stiffness for the three rotational d.o.f.s The element is typically used to couple a node of an element and a node of a guide when the element is free to move in axial direction through the guide. A shim-element material must be defined and assigned (connected) to the element.
GENERAL-SPRING-(GLSH)	General two node spring element with stiffness in all six d.o.f.s A general spring material must be defined and assigned (connected) to the element.
PILE-SOIL-(PILS)	Presently not in use.

3.3 Properties

3.3.1 Eccentricity

An eccentricity (or offset) is in effect an infinitely stiff coupling between a node and a beam end.

Eccentricities (offsets) may be defined for beam, truss and non-structural beam elements by the PROPERTY ECCENTRICITY command. Note that the PROPERTY GAP and CHANGE JOINT select-nodes GAP PLANEWISE commands also introduce eccentricities; see Section 3.6.3. If new eccentricities are to be defined using the PROPERTY ECCENTRICITY command for elements that already have been given eccentricities under a PROPERTY GAP command you must first delete the current eccentricities using the DELETE ECCENTRICITY command.

The eccentricity is given as a vector in the global, local or transformed coordinate system and pointing from the node towards the element end; see Figure 3.6.

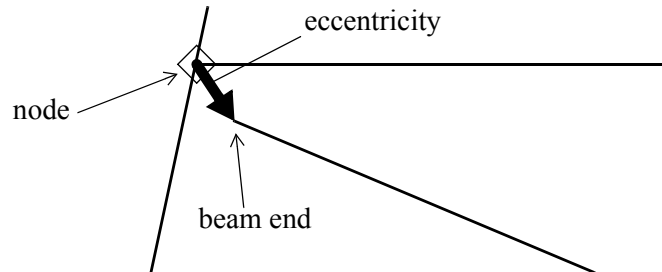


Figure 3.6 Eccentricity (or offset) is given as a vector from node to element end

Note: Introducing eccentricities for an element for which a local coordinate system has previously been defined (using the **PROPERTY LOCAL-COORDINATE** command; see Section 3.3.3) will lead to error if the eccentricity involves changing the direction of the local x-axis. This is because the y- and z-axes have been determined by the **PROPERTY LOCAL-COORDINATE** command and changing the x-axis will then give a non-cartesian coordinate system. Printing the local coordinate system for the element will reveal this by a remark (ERR-O) in the right hand column of the table.

When an element with eccentricities is subjected to a point load or distributed load the user may choose how to apply the load by the **SET ELEMENT-LOAD-DISTANCE-MODE** command; see this.

3.3.2 Hinge

Hinges may be defined for beam and non-structural beam elements. This is a generalised hinge in that any of the six d.o.f.s may individually be defined as ‘hinges’. The hinge may even have a certain resistance or a spring attached. This may be given as a value in between 0 and 1 where 0 implies fully released (hinge with no resistance) and 1 implies fully connected (no hinge).

Hinges are assigned to the two ends of the beam elements individually. A hinge is the beam end’s degree of connection to the node. Figure 3.7 is an illustration of a beam with a translational hinge in one end and a rotational hinge in the other end. Note that other elements coming into the nodes are fully connected to the nodes unless hinges are defined for these elements as well.

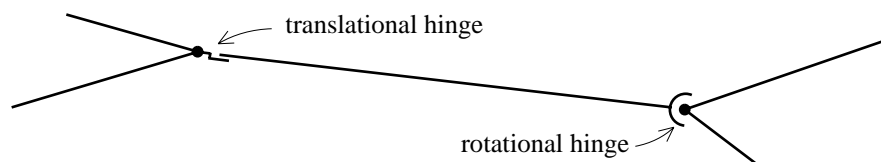


Figure 3.7 Illustration of hinges for a two node beam element

The use of hinges may easily lead to a singular or ill-conditioned stiffness matrix unless the effect of the hinges is fully understood and accounted for. The following pitfalls should be noted:

- All elements coming into a node cannot be hinged (with no or very small resistance) for the same d.o.f. as this will lead to a singular stiffness matrix (no stiffness for the relevant d.o.f. of the node).
- A beam should not be given hinges (with no or very small resistance) in both ends in such a way that a rigid body motion of the beam is allowed. This will be the case if the same translational d.o.f. in both ends are defined as hinges and likewise for the rotational d.o.f. about the element axis. A similar erroneous situation may occur for a straight line of several beams being fully connected to each other but where other beams are hinged to this line of beams (the line of beams may for instance be free to rotate about its axis).

When combining a hinge with an eccentricity, note that the hinge will in effect be at the node rather than at the beam end; see Figure 3.8.

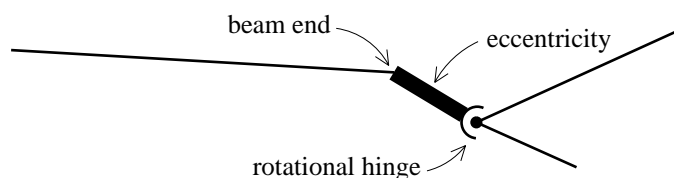


Figure 3.8 Illustration of hinges for a two node beam element with eccentricity

3.3.3 Local Coordinate System

Defining local coordinate systems are relevant for beam and non-structural beam elements only and is performed by the PROPERTY LOCAL-COORDINATE command.

The local x-axis is by definition the neutral axis of the cross section and pointing from beam end 1 towards beam end 2. Beam ends 1 and 2 are implicitly defined when creating the beam element: end 1 is the first node given when creating the element (eccentricities will, however, involve that the beam ends do not coincide with the nodes). Defining a local coordinate system involves determining the orientation of the local y- and z-axes.

Preframe will by default determine the local y- and z-axes; see the PROPERTY LOCAL-COORDINATE command for an explanation of this. Explicit definition will, therefore, only be required when this default coordinate system is not the desired one. In the case of tubular elements the orientation of the local y- and z-axes has little or no consequence and the PROPERTY LOCAL-COORDINATE command need not be used.

Note: Note that defining a local coordinate system for an element by the PROPERTY LOCAL-COORDINATE command followed by either introducing an eccentricity (by the PROPERTY ECCENTRICITY or the PROPERTY GAP commands) or changing a node position (by the CHANGE NODE command) may lead to an erroneous local coordinate system. This because the direction of the local x-axis may change while the local y- and z-axes are fixed. This will appear when printing the local coordinate systems by a remark ERR-O in the right column of the table; see the PRINT LOCAL-COORDINATE command.

3.3.4 Material

In addition to defining a linear material for the beam, truss and non-structural beam elements, the PROPERTY MATERIAL command is used to define the properties of springs and dampers. Note that the various spring and damper elements have their own type of material, i.e. not the linear material; see the PROPERTY MATERIAL command.

Further note that for all elements but the spring and damper to ground elements, the material(s) is (are) assigned to the relevant elements by the PROPERTY CONNECT MATERIAL command subsequent to defining the material(s). For spring and damper to ground elements the proper (and previously defined) material is referred to at the time of creating the elements.

3.3.5 Cross Section

Defining cross sections are relevant for beam, truss and non-structural beam elements only and is performed by the PROPERTY SECTION command. For the truss elements, however, the cross sectional data is only used to compute the cross sectional area. Any number of cross sections may be defined, each being identified by a user chosen integer number. Note that subsequent to defining the cross sections they must be assigned to the relevant elements by the PROPERTY CONNECT SECTION command.

The orientation of the cross section is defined by the PROPERTY LOCAL-COORDINATE command; see Section 3.3.3.

3.4 Align Elements

Two elements on a straight line may be defined as aligned elements by the ALIGN command. Changing the position of one of the two extreme nodes (CHANGE NODE) will then involve that the middle node moves to a new position to maintain the alignment of the two elements; see case 1 in Figure 3.9. Several elements on a straight line may also be defined as aligned by defining a chain of aligned pairs; see case 2. Finally, when two lines of elements intersect each other the elements of the two lines may individually be defined as aligned, note that the node may only be moved within the plane defined by the two lines; see case 3. Note that only the position of either of the extreme nodes may be changed.

The command may also be used to align two unaligned elements. Simply define them as aligned (even though they are not) and then go through the process of changing the coordinates of an end node.

When the switch controlling the "automatic alignment" is switched on (use the command SET ALIGNMENT-AUTOMATIC ON), elements created by the GENERATE command (i.e. jacket structure, line of elements, T-brace, K-brace and X-brace) and the SPLIT command will automatically be assigned alignment attributes.

The command ALIGN LINE can be used to add alignment attributes to all elements on a straight line between two nodes.

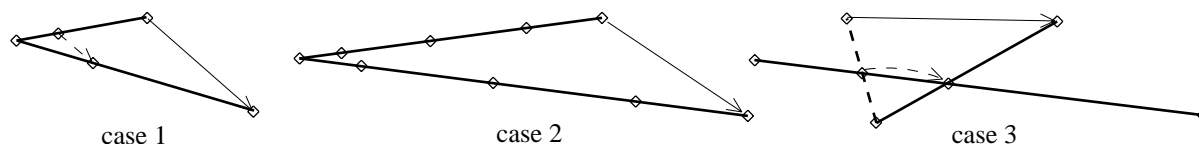


Figure 3.9 Changing a node position with aligned elements

Several inter-linked alignments (e.g. jacket legs combined with X-bracings) should be avoided. Further, note that alignments combined with eccentricities should be avoided as the existence of any eccentricities is neglected in the alignment calculations.

3.5 Members

Member concepts (segmented members) are able to hold information about several elements on a straight line between two nodes (structural joints). Special commands are used to quickly define can, stub and conical member segments in the model.

The member concept is also used to hold non-geometric information, i.e. hydrodynamic properties and stability parameters.

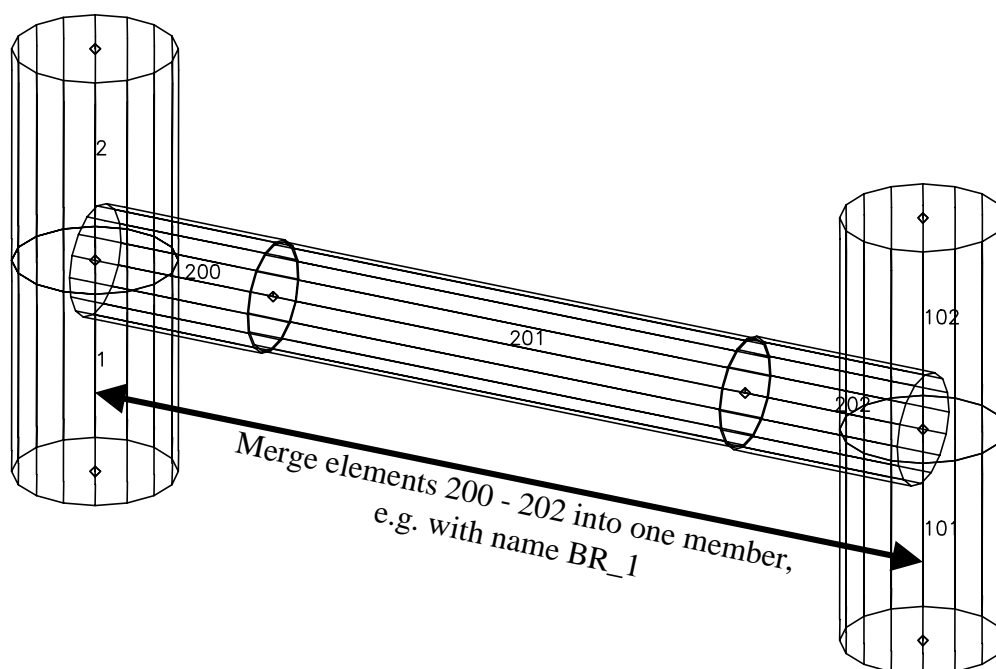


Figure 3.10 Create a member by merging elements

The member definition contains the following information:

- Element numbers between the two end nodes and in which order
- Information regarding elements representing can, stub and conical member segments
- Reference to hydrodynamic properties (Cd, Cm, flooding-status) assigned to the member
- Reference to stability parameters (buckling length, effective length factor) assigned to the member

The command `CREATE MEMBER` and `CREATE MEMBER-FROM-ELEMENTS` are used to create members. A member is a modelling concept defined by a start and an end node (joints) and one or more elements (segments) between the two nodes.

All elements belonging to a member are automatically assigned the alignment attribute. I.e. if a node in one end of a member moves, then all intermediate nodes will be moved to keep the elements forming the member on a straight line. Also, if the first or last element (segment) in a member is given an eccentricity then all intermediate elements will be applied with necessary eccentricities to keep the elements on a straight line.

See also the command `SET ALIGNMENT-AUTOMATIC`. When set to ON, elements created by the `GENERATE` command (i.e. jacket structure, line of elements, T-brace, K-brace and X-brace) and the `SPLIT` command will automatically be assigned alignment attributes.

Information related to the members is written to the SESAM Input Interface File, and the information is used by Wajac and Framework.

3.6 Tubular Joint Modelling

3.6.1 Assign joint strengthening

Define joint strengthening sections (cans / stubs) by use of the command `ASSIGN CAN` and `ASSIGN STUB`.

The `ASSIGN` command assigns / updates members with new elements. Additional elements are added to a member in order to represent a can or stub section at one end of the member. Adding an element in this connection actually means splitting an existing element. This operation presumes that the element to be modified is part of a member, and if not, the user is guided through the process of creating a member after other necessary input has been given.

The default can and stub lengths for tubular joints are calculated according to predefined geometric rules. The rules are defined by the parameters defined in the command: `SET CAN-STUB-LENGTH-PARAMETER`.

Can and stub sections can only be the start or end segment of a member, hence members shall normally be defined between two "structural joints". The member information is read and used by Framework.

Two switches can be used in connection with the `ASSIGN` command:

- `SET ASSIGN-OPTION MANUAL-NUMBERING`; when switched ON it allows the user to manually give node and element number to the node and element created by the command.
- `SET ASSIGN-OPTION SECTION-NUMBER`; when switched ON it allows the user to give a section number to the strengthening element instead of specifying PIPE geometry data.

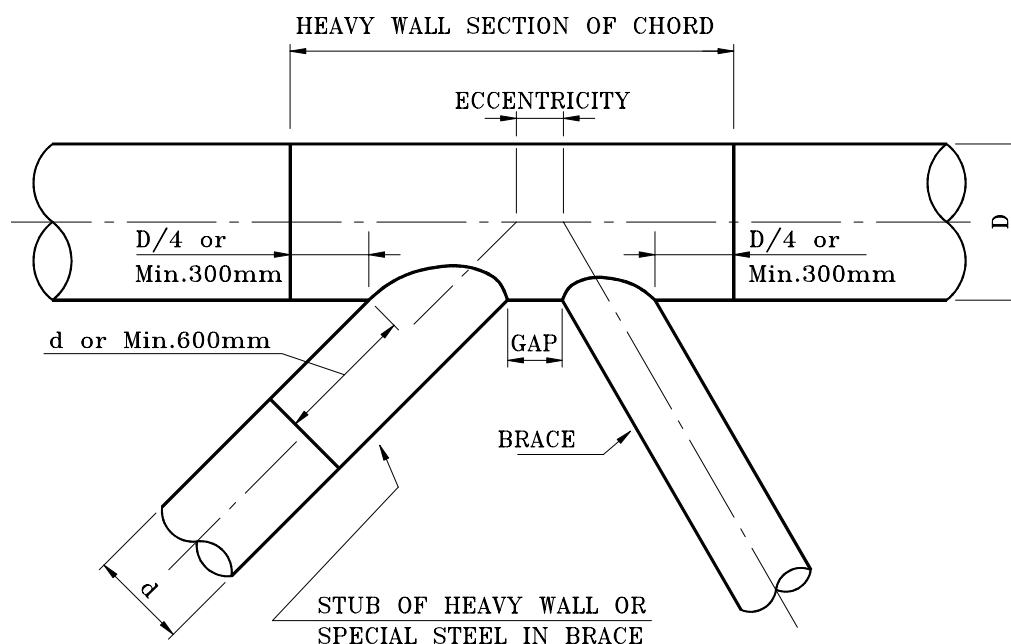


Figure 3.11 Detail of simple joint with strengthening

3.6.2 Change joint geometry

The command CHANGE JOINT is used to update joints regarding required length of can and stub sections, and for calculation of brace eccentricities to satisfy minimum gap between braces (planewise, i.e. plane defined by chord and braces).

Update joints with brace eccentricities due to minimum gap by use of the command CHANGE JOINT select-nodes GAP-PLANEWISE. (See Section A 3 for illustrations of the effect of this command.)

Update can and stub lengths due to change in joint lay-out by use of the command CHANGE JOINT select-nodes CAN-STUB-LENGTH.

Similar as when first defined, the can and stub lengths for tubular joints are calculated according to predefined geometric rules. The rules are defined by the parameters defined in the command: SET CAN-STUB-LENGTH-PARAMETER.

The commands CHANGE CAN and CHANGE STUB can be used to change both pipe section and lengths of joint strengthenings.

3.6.3 Advanced gap calculations, command PROPERTY GAP

Structures consisting of tubular beams welded together in tubular joints constitute a most common type of design. In the joint the tubular members with smaller diameters, termed braces, are welded onto the member with the largest diameter, termed the chord. A preferred design of such tubular joints is to have a certain minimum gap between the intersections of the various braces and the chord. Therefore, if two braces over-

lap or their intersections with the chord are too close then the intersections are moved apart to ensure a proper gap. The effect of this is that the neutral axis of one or more of the braces will not pass through the node. This may be modelled as eccentricities or offsets of the brace ends compared to the nodes.

A PROPERTY GAP command is available for automatic introduction of eccentric attachment of braces (beam elements) to joints (nodes). The brace ends are moved in the chord x-axis direction away from the node to ensure a proper gap between the brace-chord intersections. Brace ends are moved only when the gap is less than a minimum value (see SET LEGAL-GAP). The calculation starts by moving the brace closest to the ‘fixed brace’ (see below), then fixing that brace and moving the next brace and so on. Note that the chord actually consists of two aligned beam elements, the one with the largest diameter is termed chord and the other is termed aligned. If their diameters are equal (which often is the case) the lowest element number is the chord.

The PROPERTY GAP command has the following appearance:

PROPERTY GAP node chord-el aligned-el legal-gap ...

...	FIXED-BRACE	element-no			...
	IGNORE-BRACE	element-no	element-no		
	SYMMETRIC-ELEMENTS	element-no	element-no		

...	FIXED-BRACE	element-no		
	GAP	element-no	element-no	legal-gap
	END			

If an eccentricity has been specified for a brace prior to giving the PROPERTY GAP command, the eccentricity will be maintained unless the gap is less than the minimum value. I.e. if the gap is larger than the minimum value the eccentricity previously specified will not be changed.

See the warning in Section 3.3.1 about introducing eccentricities for an element for which a local coordinate system has already been defined (the PROPERTY GAP command introduces eccentricities).

PROPERTY GAP in graphical user interface

The DISPLAY FOOTPRINT command is convenient in combination with the PROPERTY GAP command. The use of the PROPERTY GAP command subsequent to a DISPLAY FOOTPRINT command is illustrated below. Use of the graphical user interface — see Section 3.1 — is assumed here but the functionality is the same for line-mode.

Having displayed the footprint of a joint a screen display as shown in Figure 3.12 appears. A part of the chord is presented in developed view with the brace intersections shown. The quadrilateral formed by the horizontal broken lines and the vertical solid lines represents the developed chord. The node number as well as the chord element number and aligned element number are given in the upper right corner. (Preframe determines the chord and aligned elements based on the largest diameter of the tubes coming into the joint.)

Upon giving the PROPERTY GAP command you are requested for joint (node), chord and aligned element. You may enter the numbers by the keyboard or selected them by clicking the left mouse button on the numbers in the upper right corner of the screen as indicated in Figure 3.12.

Thereafter, the 'legal gap' should be given, i.e. the minimum allowable gap between brace-chord intersections. Preframe suggests a value that you may accept by hitting Return, or override by another value. You may also change the suggested default value using the SET LEGAL-GAP command prior to the PROPERTY GAP command.

The method of moving the brace ends is based on that the brace(s) perpendicular to the chord is (are) defined as fixed, i.e. it is (they are) not moved while the other braces are moved away to ensure proper gaps. When no braces are perpendicular to the chord then no braces are defined as fixed. Instead, the two braces closest to and on each side of a plane perpendicular to the chord are suggested as 'symmetric elements'. These are moved the same distance in opposite directions to ensure a proper gap. The other braces are moved away from the symmetric elements to ensure proper gaps all through the joint.

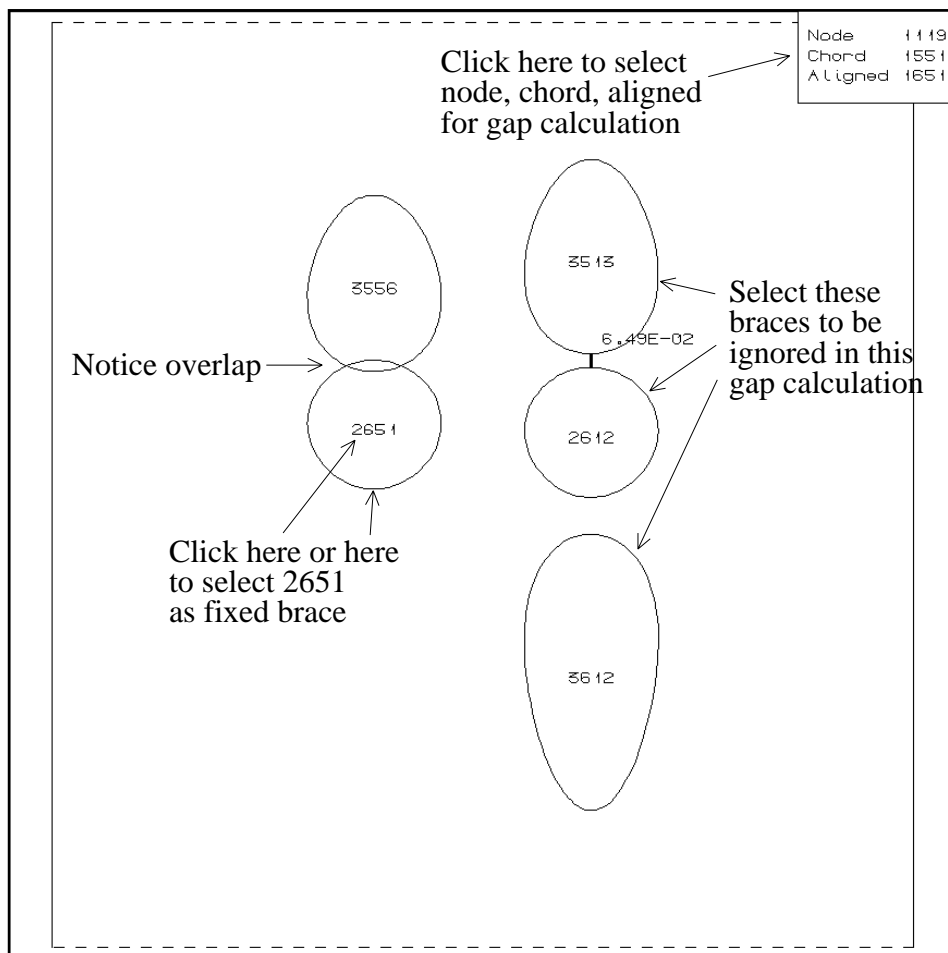


Figure 3.12 The footprint display before the gap calculation

The next step is therefore to define the 'fixed brace(s)' and/or 'symmetric elements'. Preframe determines which brace is closest to being perpendicular to the chord element and suggests this one as the 'fixed brace'. You may choose this brace by hitting Return or give another brace element number. If you want to define more elements as 'fixed braces' then repeat the FIXED-BRACE subcommand and element number for all these elements before you close the PROPERTY GAP command by the END alternative.

If no braces are close to being perpendicular to the chord, then the two braces closest to and on each side of the perpendicular plane are suggested as 'symmetric elements'. Either a 'fixed brace' or two 'symmetric elements' must be defined.

Selecting elements to be 'fixed braces' or 'symmetric elements' may also be done by clicking the mouse on the element numbers or the elements' intersection curves within the screen display as indicated in Figure 3.12.

There may be some braces which are to be left out of the current gap calculation, i.e. not moved. (This may for instance be because they will be, or has been, taken care of in another PROPERTY GAP command.) Before defining the 'fixed brace(s)' or 'symmetric elements', the IGNORE-BRACE subcommand should be used to eliminate these from the calculations.

A gap value different from the 'legal gap' (given in the beginning of the PROPERTY GAP command) may be given for a pair of braces. Use the GAP alternative followed by the relevant pair of element numbers to achieve this. For example, specifying a gap of 0.0 between a pair of braces allows these to overlap while other braces are moved to ensure proper gaps between this pair and other braces.

The PROPERTY GAP command is closed by the END alternative after having defined IGNORE-BRACE, FIXED-BRACE and/or SYMMETRIC-ELEMENTS. Preframe will then calculate the required eccentricities (offsets) and display the result; see Figure 3.13.

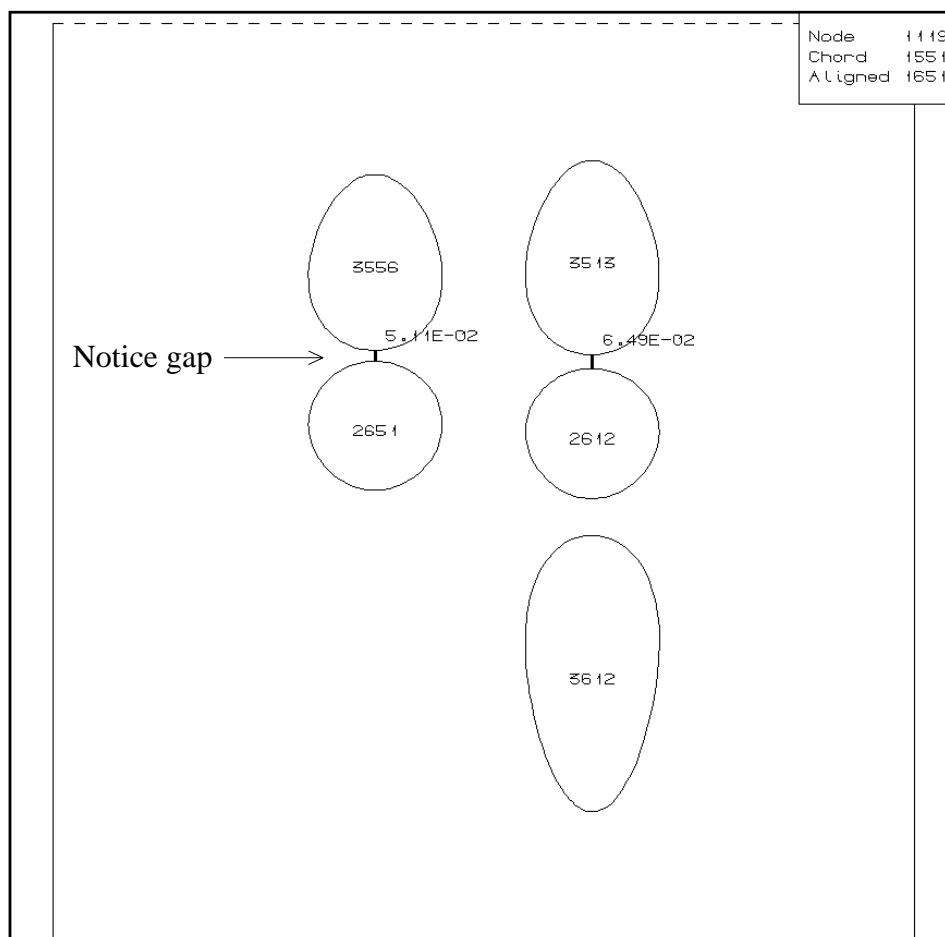


Figure 3.13 The footprint display after the gap calculation

There is a set of gap values used in the PROPERTY GAP and DISPLAY FOOTPRINT commands. These are:

- legal gap,
- display gap,
- zero gap.

all having default values set by the program and which may be altered by the SET command. The 'legal gap' is used as default value in the PROPERTY GAP command as described above. See the DISPLAY FOOTPRINT command for an explanation of the use of the 'display gap' and 'zero gap' values.

Note that eccentricities (computed by the PROPERTY GAP command or specified directly) are copied along with elements. This is convenient as limiting use of the PROPERTY GAP command saves both manual work and computing time (the command is somewhat resource demanding). However, when using the GENERATE command to create a jacket as explained in Section 3.2.4 the whole jacket with all its joints may have been created without taking advantage of introducing gaps only once for identical joints. The

PROPERTY GAP command may therefore have to be used for all joints individually. This problem may be reduced by either or both of the following approaches:

- Create (or refine) the bracing in only one panel, use the PROPERTY GAP command to introduce eccentricities and then copy the panel. Elements already existing in the destination, e.g. elements in the leg and horizontal bracing, will not be overlapped.
- Establish a command input file containing the PROPERTY GAP commands and execute these commands in non-interactive mode.

3.7 Hydrodynamic and Stability Data

Hydrodynamic and some stability data may be assigned to member concepts. In Preframe this is called conceptual attributes.

The command ASSIGN HYDRODYNAMIC is used to assign hydrodynamic coefficients to selected members. The following coefficients may be assigned:

- Drag coefficient
- Inertia coefficient
- Flooding parameter (i.e. flooded or non-flooded when immersed)

The command ASSIGN STABILITY is used to assign stability buckling parameters to selected members. The following parameters may be assigned:

- Effective buckling factor
- Buckling length

This conceptual information related to members is written to the SESAM Input Interface File, and relevant information is read by Wajac and Framework.

3.8 Soil and Pile Modelling

Modelling of soil profile and piles (pile concepts) is available in Preframe. Special input commands are used to quickly define the soil profile and pile geometry inclusive pile attributes. Hence, a combined model of the jacket structure and the soil / piles may be created. Information related to the pile concepts (geometry and attributes) are written to the SESAM Input Interface File, and the pile data are read by the non-linear pile-soil interaction analysis program Splice. Modelling of piles in Preframe implies that use of the pile data generation program Pilgen is not necessary prior to running Splice.

Pile definitions and soil data together forms the basis for Preframe to write an input file (template) to the Splice analysis.

The data given with respect to the soil profile may be used as basis for generation of piles, i.e. the piles will be generated to match the soil layers. Based on the soil data, Preframe writes an input file (template) to be used by the soil stiffness calculation program Gensod.

The pile elements are part of the first level superelement, and hence may be handled in the postprocessing program Framework (e.g. print / display of element forces, print of stresses, display of shape, yield code check of the pipe elements, etc.). For example, see Appendix A.3.

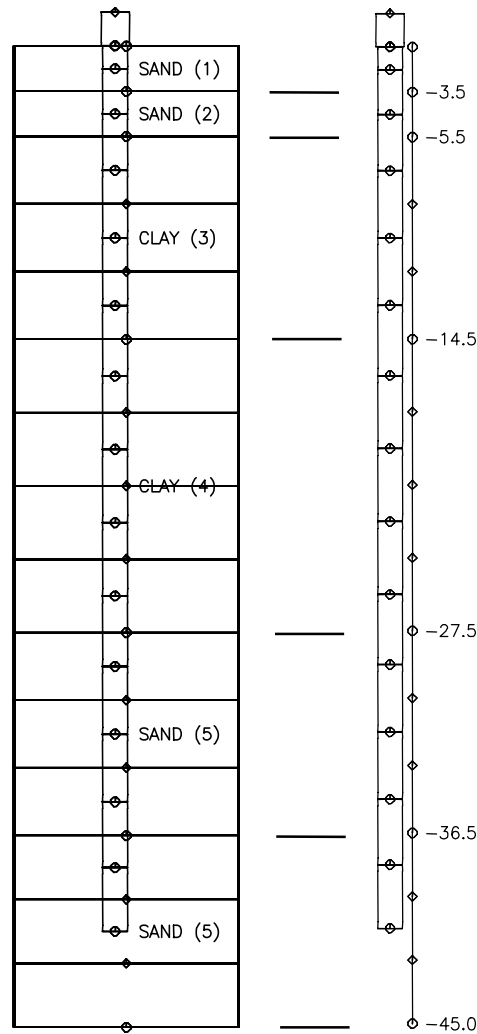


Figure 3.14 Model of single-pile showing soil profile and Z-level for each soil type

Data definitions describing the soil profile and data types exist for:

- soil types
- soil profile with respect to soil types and layer divisions
- skin friction and tip resistance data
- PY, TZ and QZ codes

Use of soil (and pile) modelling presumes that the Preframe model is defined with global Z axis pointing upwards.

Generate necessary data in the following order:

- 1 Generate the jacket structure. (Global Z axis pointing upwards.)
- 2 Define the soil data, i.e.
 - mudline level
 - soil types (sand / clay)
 - soil profile (soil types and layer divisions)
 - skin friction and tip resistance data
 - PY, TZ and QZ codes
- 3 Generate the piles.
 - generate piles based on soil profile or user given segment lengths and number of elements
 - add necessary pile data (attributes), i.e. yield strength, tip-code, "fixed-to" node reference when pile groups, density of fluid inside piles
- 4 By use of the WRITE command, create the:
 - SESAM Input Interface File.

The superelement which contains the piles must be number 1, i.e. T1.FEM.
 - Gensod input / template file
 - Splice input / template file

3.9 Boundary Condition

By default, the boundary condition code for all d.o.f.s of all nodes is 'free'. Using the BOUNDARY command one of the following boundary condition codes may be given to selected d.o.f.s for selected nodes:

- free
- fixed
- prescribed
- super

For a d.o.f. defined as 'prescribed' the prescribed displacement must subsequently be given by the LOAD load-case NODE PRESCRIBED-DISPLACEMENT command, alternatively PRESCRIBED-ACCELERATION.

When one or more super d.o.f. have been defined the superelement technique is implicitly being applied and the model created cannot constitute a complete model. It is by definition a first level superelement that must be assembled using Presel (as the only superelement or more likely together with other superelements) to form the complete model as a higher level superelement.

3.10 Change Data

Most data defined may be changed using the CHANGE command. This is relevant for instance when erroneous data has to be corrected.

The following two sections describe in more detail two of the many commands for changing data. These two commands have special effects that may be utilised in situations other than for correcting erroneous data.

3.10.1 Change Nodal Coordinates

In addition to changing the nodal coordinates in the same manner as they were defined by the NODE command a group of nodes may be changed by the following two commands:

- CHANGE NODE TRANSLATE
- CHANGE NODE ROTATE

The former command will translate the selected nodes according to a vector given.

The latter command refers to a transformed coordinate system defined by the TRANSFORMATION command. The procedure is illustrated in Figure 3.15 (the shaded areas represent the selection of nodes that are moved, all in the XY-plane) and can be described as follows:

- the positions of all selected nodes are calculated in the transformed coordinate system,
- these new coordinates values are then interpreted in the model's (global) coordinate system.

The effect of this procedure will in the illustrated example be that the nodes are rotated in the direction from the transformed coordinate system towards the global coordinate system.

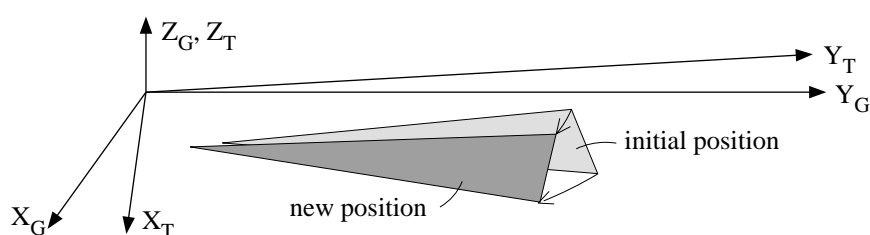


Figure 3.15 Changing node positions by CHANGE NODE ROTATE

3.10.2 Change Load to Mass

Dead weights may have been modelled as vertical forces (in negative z-direction) for a static analysis. If a dynamic analysis is to be performed these must be converted to nodal masses. Preframe offers a command for this purpose, the CHANGE LOAD load-case TO-MASSSES command.

Within the command reference is made to a gravity load case. The transformation of forces to masses is performed by dividing the forces by the acceleration of gravity taken from the referred gravity load case. Unless the forces and the acceleration of gravity are parallel and with the same sign (normally both will be in negative z-direction) the command will refuse to change the loads into masses.

Note that the weight of nodal masses, see Section 3.13, will contribute in a gravity load case in a static analysis. Dead weights may therefore be modelled as masses rather than nodal forces thereby letting them contribute to both static and dynamic analyses.

3.11 Copy Data

The COPY command may copy either a line segment or a plane of nodes and elements. The line segment to copy is selected by referring to its extreme nodes and its new position is defined by giving two nodes that previously must have been created. The plane to copy is selected by referring to three nodes in the plane (not necessarily extreme nodes) and its new position by giving three nodes that previously must have been created. The triangle formed by the three nodes to be copied must be congruent with the triangle formed by the three nodes of the new position. The line segment to copy may be contracted or extended.

Most, but not all data assigned to the elements are copied along with the elements; see Section 5.1 for details.

The copy command will never duplicate nodes or elements.

Note: Member concepts are not be copied. When copying elements and nodes which are part of a member concept, new elements and nodes will be created, but no conceptual information will be copied.

3.12 Linear Dependency

The LINEAR-DEPENDENCY command offers two alternative types of linear dependency:

- general node dependency,
- two node dependency.

In the former any d.o.f. of a node (the dependent d.o.f.) may be made linearly dependent on any other d.o.f.s of any other nodes (the independent d.o.f.s). The user explicitly specifies the linear dependency factor for all the independent d.o.f.s. The displacement of the dependent d.o.f. will then be:

$$r_d = r_{i1} \times b_1 + r_{i2} \times b_2 + r_{i3} \times b_3 + \dots$$

where r represents the displacements, subscripts d and i represent the dependent and independent d.o.f.s respectively and b is the dependency factors given.

With the 'two node dependency' all d.o.f.s of a given node are made linearly dependent on the corresponding d.o.f.s of two other nodes. The displacement of the dependent d.o.f.s will be:

$$r_d = r_{i1} \times b + r_{i2} \times (1 - b)$$

where b is a dependency factor given by the user. Preframe will compute a default value for b as explained in Figure 3.16. b is computed based on the projection of the dependent node (dependent nodes are indicated by blue triangles on colour screens) onto the line between the two independent nodes (independent nodes = supernodes are indicated by blue octagons).

Normally, the 'two node dependency' has physical meaning only when the dependent and the two independent nodes lie on a straight line.

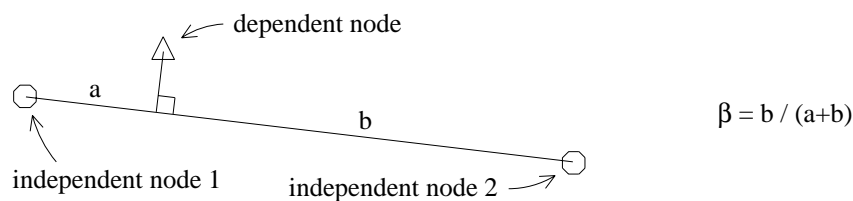


Figure 3.16 Two node linear dependency - the dependency factor b

Note: All independent d.o.f.s must be super d.o.f.s, i.e. they have to be defined with boundary condition code 'super' using the BOUNDARY command prior to giving the LINEAR-DEPENDENCY command. (The 'two node dependency' alternative allows the user to define the independent nodes as super within the command by the FORCE-INTO-SUPER alternative.) This implies that using linear dependency involves using the superelement technique, also see Section 3.9.

The only exception from the above Note is if the option SET LINEAR-DEPENDENCY-MODE NO-FORCE-TO-SUPER is used. It is then possible to use linear dependency between nodes without use of super nodes (i.e. without use of the super elements technique). This option is only available when using the multifront equation solver in Sestra.

Linear dependencies in a transformed coordinate system may be specified by first assigning a transformation to the dependent and independent nodes using the BOUNDARY command.

3.13 Nodal Mass

The MASS-ON-NODE command allows concentrated masses (a diagonal mass matrix) to be defined for selected nodes. Note that the weight of nodal masses will contribute in a gravity load case in a static analysis.

3.14 Transformation

Transformations are defined by the TRANSFORMATION command and are used for the following purposes:

- to introduce rotated coordinate systems in nodes for the purpose of defining boundary conditions not coinciding with the global axes, e.g. fixed in a certain direction and free in the two perpendicular directions,

- to define the local coordinate systems of elements with no extent, e.g. spring-to-ground elements,
- to define loads in coordinate systems other than the global (the model's coordinate system),
- to rotate groups of nodes; see Section 3.10.1.

3.15 Load

Loads are defined by the LOAD command and should be given consecutive load case numbers (1, 2, 3, ...) or else computation time will be spent in the analysis program solving zero load cases. The following types of loads may be given:

- element load,
- nodal load,
- gravity,
- rotation of structure.

For more information see the LOAD command.

Note: Element loads are removed when elements are modified by the ASSIGN and SPLIT commands, i.e. loads (element loads) are applied to elements which by these commands are being deleted. Hence, complete the modelling prior to defining load cases including element loads. (By use of the command LOAD load-case ELEMENT LINE-LOAD node1 node2, element distributed loads will be created on all elements between the two given nodes.)

3.16 Display Features

See Section 4.1.4 for general information on requirements to the graphics environment.

There are numerous features for displaying the model. The relevant commands are explained in the following.

DISPLAY displays the model. Its subcommands are:

- **NODE**
displays the nodes and elements; see Figure 3.17.
- **ELEMENT**
displays the elements only, i.e. without nodes; see Figure 3.18.
- **JOINT**
displays a joint with all elements — chord and braces — meeting there, the elements are displayed in facet draw mode together with the intersection curves between them, the current eccentricities are accounted for; see Figure 3.19. In graphical user interface, selection of both elements and nodes is available by clicking the mouse. For example, give the command DISPLAY JOINT and click the mouse at the free end of

one of the elements to display the neighbouring joint. Or click at the joint itself within the DISPLAY FOOTPRINT command.

The LABEL command cannot be used for a display of a joint. The commands ROTATE, SET GRAPHICS and ZOOM are allowed.

The remarks below for the DISPLAY FOOTPRINT command on how the chord is determined and assumed to be continuous through the node and that the chord-brace intersections are between the outer surfaces, are valid also for the DISPLAY JOINT command.

- **FOOTPRINT**

displays the footprint of a joint, i.e. a developed view of a part of the chord with the brace intersections shown; see Figure 3.20. The quadrilateral formed by the horizontal broken lines and the vertical solid lines is the developed chord. The element numbers of the braces are given in the middle of the intersection curves. The gaps between the intersections will also be displayed with their current values. The gaps are only shown when they are larger than the 'zero gap' (see SET ZERO-GAP) and smaller than the 'display gap' (see SET DISPLAY-GAP). Yellow colour is used for gaps larger than the 'legal gap' (see SET LEGAL-GAP) and red for gaps less than the 'legal gap'. Overlaps are treated in the same way as 'zero gaps' i.e. they are not shown.

Note that the chord is determined based on the largest diameter of the tubes coming into the joint. And in calculating the chord-brace intersections the chord element is assumed to be continuous through the node past the last chord-brace intersection and irrespective of whether the aligned element has a smaller diameter than the chord, or whether it exists at all.

Also note that the intersection curves are the intersections between the outer surface of the chord and the outer surfaces of the braces.

Note that the gap is calculated as the distance in space (not along the chord outside) between the two intersection curves. The gap is the smallest distance found.

The commands LABEL, ROTATE, SET GRAPHICS and ZOOM cannot be used for a display of a footprint.

RE-DISPLAY

re-displays the last display without any labelling.

LABEL

annotates node symbols, node numbers, etc. to the displayed model.

- **NODE-SYMBOLS**

The following symbols are used:

- yellow diamond, i.e. all six d.o.f. free or fixed
- blue triangle, i.e. one or more d.o.f. linearly dependent
- blue octagon, i.e. one or more d.o.f. super

Using the DISPLAY NODE command the node symbols are shown automatically.

- **NODE-NUMBERS**

- **ELEMENT-NUMBERS**

- **SECTION-NUMBERS**

- **LOCAL-COORDINATE**

adds either the local y- or z-axis for all elements.

- **MATERIAL-NUMBERS**

- **BOUNDARY-CONDITION-SYMBOLS**

A set of blue symbols illustrates the boundary conditions applied; see the LABEL command for details.

- **ORIGIN-SYMBOL**

adds the origin in its correct position to the display.

PLOT

generates a plot file of the last display or of the complete model. The plot file should be sent to a plotter or laser printer.

ROTATE

rotates the display of the model. The SET GRAPHICS EYE-DIRECTION command is an alternative command for changing the view point.

SET GRAPHICS

sets and defines various control parameters for the DISPLAY, PLOT and LABEL commands.

- **AUTO**

The display will automatically be updated when new nodes and elements are added, this is default for graphical user interface.

- **DEVICE**

sets the proper type of graphics device.

- **EYE-DIRECTION**

sets the desired view point.

- **HIDDEN**

switches to hidden mode.

- **INPUT**

switches between line-mode (relevant for workstations only) and graphical user interface.

- **PRESENTATION**

sets the desired draw mode for tubular beam elements: 'facet', 'silhouette' or 'wire-frame', the last one is the default. In Figure 3.17 the 'wire-frame' mode is used. In Figure 3.21, the 'silhouette' mode is shown. The 'wire-frame' mode is suitable for displaying the complete model while the 'silhouette' mode is suitable for a plane showing the sizes of the pipes. The 'facet' mode is suitable for details and it also used in the DISPLAY JOINT command.

- **SHRINK-FACTOR**

enables a display with elements shrunk in size.

- **SIZE-SYMBOL**

sets the sizes of a number of symbols annotated by the LABEL command.

- **ZOOM**

increases or decreases the scale of the display. To zoom give the ZOOM IN/OUT command, point to a corner of the zoom area, press and hold the left mouse button, drag the pointer to the opposite corner and release the button. Figure 3.22 shows how a box is made while dragging the pointer over the zoom area. Figure 3.23 shows the result. In this display the chord-brace intersection curves are clearly seen. The DISPLAY NODE/ELEMENT commands will revert to a default scale fitting the complete model within the frame.

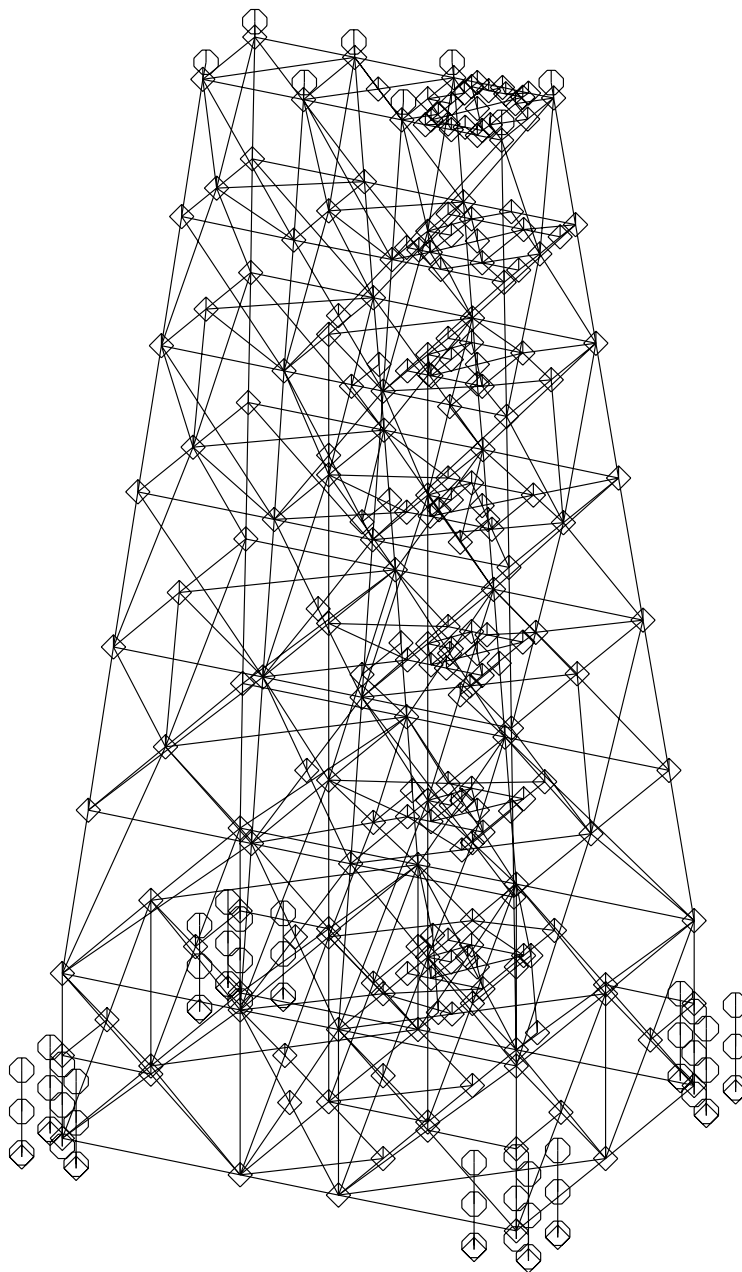


Figure 3.17 The result of the **DISPLAY NODE** command

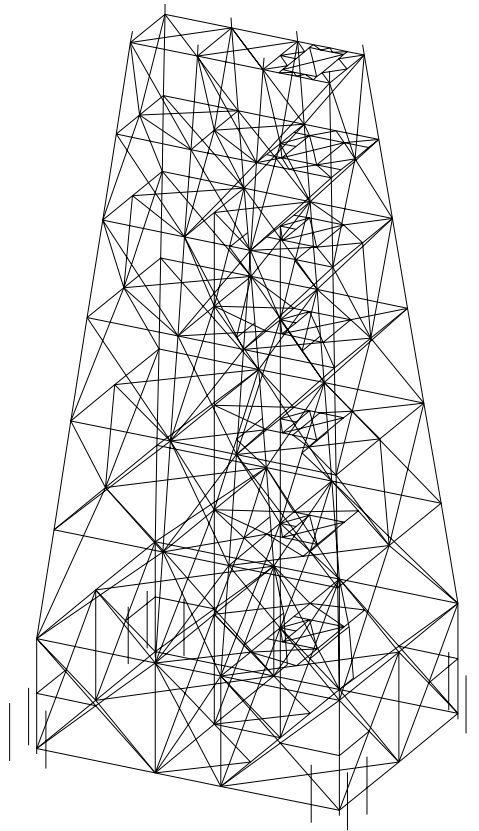


Figure 3.18 The result of the **DISPLAY ELEMENT** command

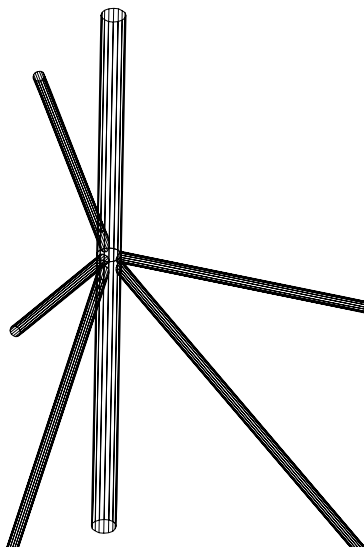


Figure 3.19 The result of the **DISPLAY JOINT** command

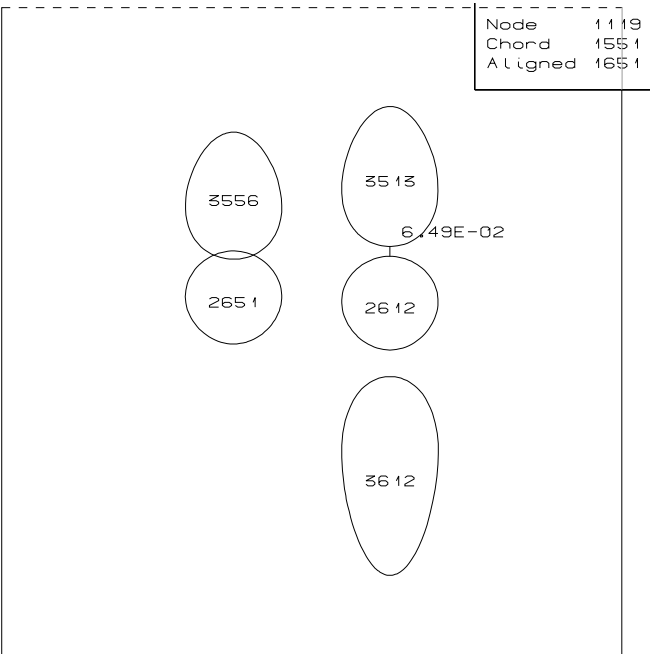


Figure 3.20 The result of the DISPLAY FOOTPRINT command

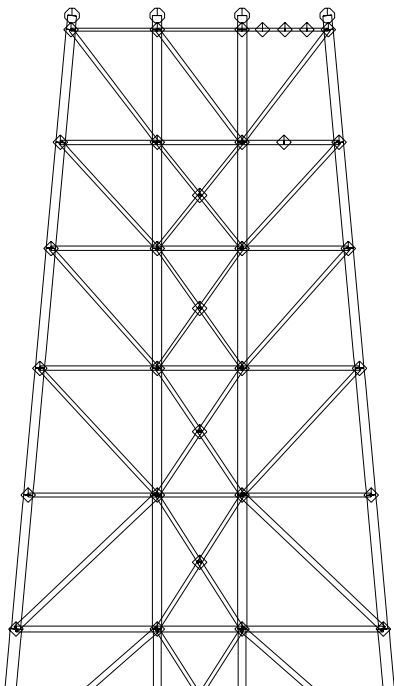


Figure 3.21 The result of the DISPLAY NODE PLANE command with 'silhouette' draw mode

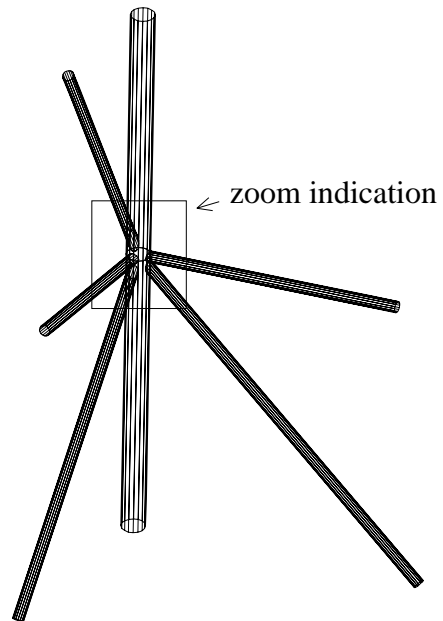


Figure 3.22 Zooming in on the display of a joint

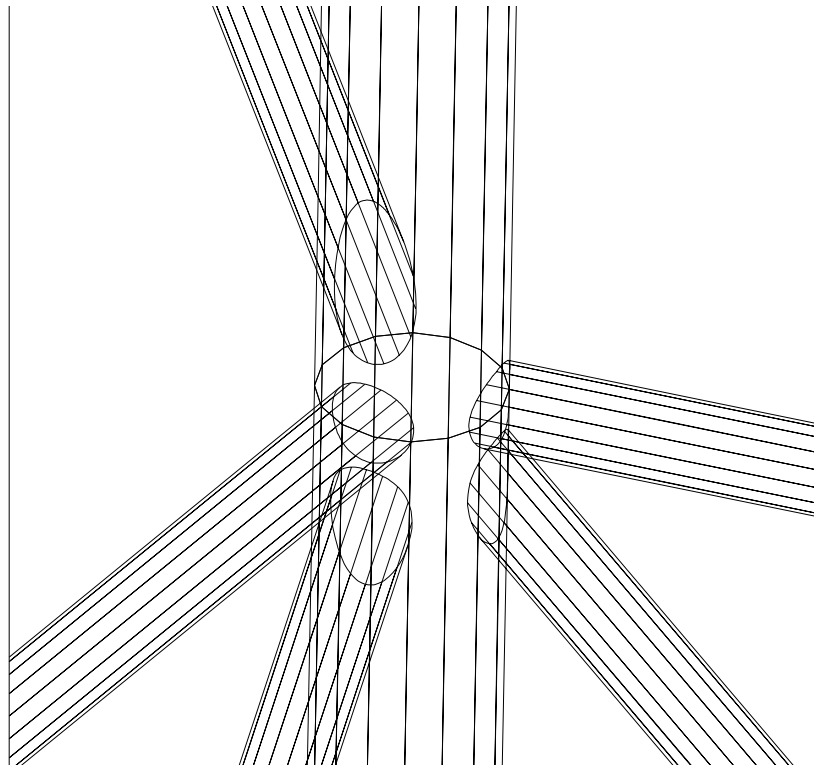


Figure 3.23 The zoomed in joint

3.17 Printing Data

The PRINT command enables printing of any data entered by the user or computed by the program. By default the PRINT ALL command — which will produce a full report of the model — will be directed to a print file. All other PRINT commands will by default produce the data on the screen (the line-mode window for workstations and print window for Windows NT).

The SET PRINT command may redirect the tabulated data, set the name of the print file and change the page size (number of lines per page).

3.18 Writing input templates to soil stiffness and pile-soil analyses

The command WRITE GENSOD-SPLICE-TEMPLATE writes the input files / templates to be used by the soil stiffness calculation program Gensod and the soil-pile-interaction analysis program Splice. Using this command presumes that soil profile and piles have been defined in the Preframe model.

3.19 Writing end-cut data for the jacket installation program

The command WRITE END-CUT-DATA is used to write endcut data for brace to chord connections to separate file for use in the Installjac launch program.

3.20 Writing and Reading Input Interface File

The WRITE command transfers the model to the Input Interface File. The READ command reads an Input Interface File into Preframe. See Section 2.5.

4 EXECUTION OF PREFRAME

This chapter provides information on:

- How to start Preframe
- How read a Command Input File into Preframe
- How to execute Preframe outside Manager, Unix only
- Line-mode input syntax
- Files used
- Creating plots for reports
- Alternative execution modes
- Program requirements
- Program limitations

4.1 Program Environment

Preframe is available in the following hardware environments:

- Unix computers of various vendors
- Windows 98, NT and 2000, often referred to as PC

4.1.1 Starting Preframe from Manager

Preframe is started from Manager by clicking **Model | Frame Preframe**. The graphical user interface of Preframe is explained in Section 3.1.

On Unix the graphical user interface is based on OSF/Motif X Window System.

4.1.2 Reading a Command Input File into Preframe

In the Frame Modelling window opening up when giving Model | Frame Preframe in Manager there is a box for specifying a Command input file; see Figure 4.1. By default this is set to None. Changing this to File name a new box appears in which you may specify a Command input file that will be automatically read into Preframe once the program is started by clicking OK. If the box Run interactively after command input file processing is checked Preframe will display the geometry model created by the input file and await interactive user input. You may then continue modelling or only verify the current model and leave Preframe by the EXIT command.

Note: If your Command input file constitutes the complete input then make sure the Database status is set to New. You need to change Old to New if you previously have run Preframe in which case there will exist a Preframe database causing the Database status to come up as Old.

Note: On the other hand, if your Command input file shall be added to an existing model then leave Database status as Old. In this way you may repeatedly add Command input files to build up your complete model. You may for instance first read a file containing definition of your preferred beam cross section types, leave Preframe and then read the modelling input referring to these cross section types.

Note: You may also read a Command input file from inside Preframe by using the SET COMMAND-INPUT-FILE command followed by the # command; see these.

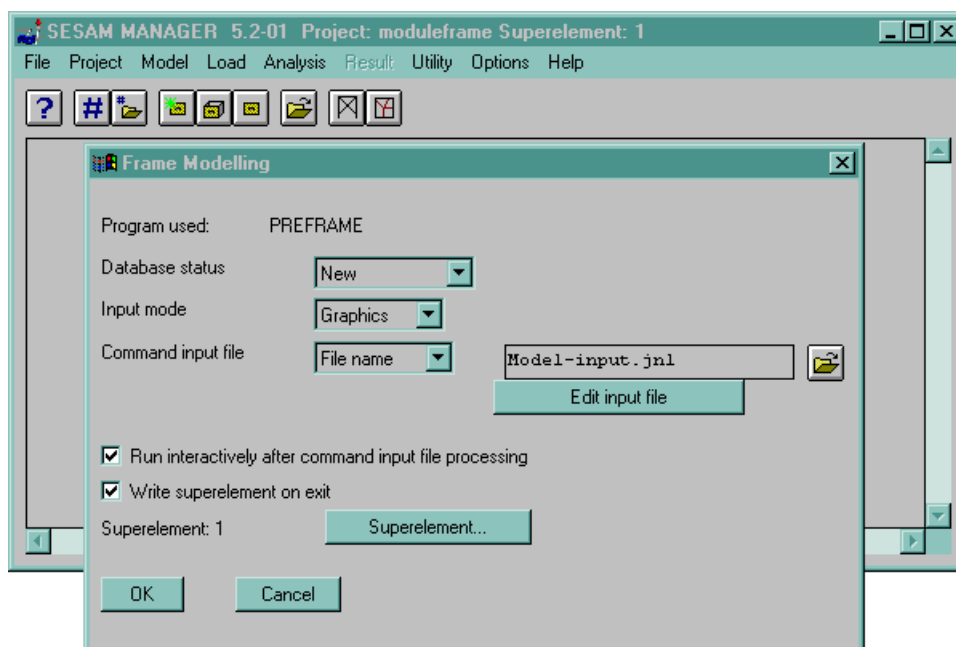


Figure 4.1 Manager and the Frame Modelling window with Command input file specification

4.1.3 Starting Preframe as an Individual Program on Unix

Alternatively to starting Preframe from Manager it may on Unix be started as an individual program. Provided SESAM is properly installed on your computer Preframe is started by the command 'preframe' (in lower case).

The program responds by presenting itself and giving some key information:

- Program version number and release date (date of executable file)
- Access date and time (now)
- Your user identification and other operating system and hardware related data

Before you are allowed to continue, i.e. start giving commands, you must respond to the following requests for information:

- 'General file name prefix'
A character string forming a part of (by preceding) the names of the files opened by the program. It may and may not include a directory specification.
- 'Model file name'
The name of the 'model file' (a binary data base file), the 'command log (journal) file' and other files opened by the program; see Section 4.1.5.
- 'Old or new file'
Choose 'new' when commencing modelling and 'old' when continuing an earlier session, i.e. the 'model file' and 'command log file' already exist.
- 'Superelement type'
Or 'superelement number', the identification or name of the superelement (model) to be created. This question will not appear when answering 'old' to the previous question.

4.1.4 Line-Mode Input of Commands and Arguments

Having successfully entered Preframe as explained in Section 4.1.3 the command prompt appears, the '#' character. The window in which you are now working is called the line-mode window. You may switch to the graphical user interface explained in Section 3.1 by the line-mode command SET GRAPHICS INPUT ON or you may continue working in the line-mode window (displaying the model will then open a display window).

The information below is about entering line-mode commands. Note that line-mode commands may also be entered in the graphic-mode window. The information below is, with a few exceptions, therefore also relevant for the graphical user interface.

The syntax and characteristics of line-mode input are as follows:

- The parameters (commands, sub-commands and data) are separated by one or more blank characters (or a comma) and may be entered one by one or with two or more entries on a single line of input. For example:

```
COMMAND
SUB-COMMAND
SUB-SUB-COMMAND
data ...
```

is equivalent to:

```
COMMAND SUB-COMMAND SUB-SUB-COMMAND data ...
```

Note, however, that data belonging to different data sets cannot be entered on a single line.

- UPPER CASE = lower case (all commands will be logged on a 'command log file' in UPPER case).
- Commands and sub-commands may be abbreviated as long as they are unique. In a command consisting of words separated by hyphens, each word may be abbreviated or completely left out. Examples:

```
NODE-NUMBERS = N-N
COMMAND-INPUT-FILE = C-I
```

- Default values are provided between slashes, '/default/'. The defaults are accepted by hitting Return.
- Real or integer input may be entered irrespective of type of numerical data, use 'E' for exponent.
- '?' will list all legal commands and data options. (This command is irrelevant for the graphical user interface where all legal commands and data options are at any time given in the command column of the graphic-mode window.)
- 'P?' will list all legal commands starting with P.
- '..' (two dots) will execute the input data before '..' and subsequently abort the current command. The program is thereafter ready for more commands. If the data before the '..' is incomplete it will be discarded.
- ',,' (two commas) will cause one default parameter to be accepted. (May be useful when editing a 'command input file'.)
- ';' (semicolon) will cause default parameters to be accepted until the end of the parameter group or until there is no default provided.
- Text containing blank characters has to be enclosed within single quotes: 'this is a text'.
- '%' (percentage sign) at the beginning of a line is used for entering a comment. Comments will be logged together with commands on the 'command log file' (see Section 4.1.5). Note that the program will occasionally log information on the 'command log file', this will appear as comments in between data and comments entered by the user. The program information is preceded by '%%' (two percentage signs) to distinguish it from the user's own comments. This makes it easy to strip a 'command log file' for program information in connection with creating a 'command input file' (any fairly good editor will have a macro-functionality or similar enabling you to locate and remove all lines with '%%'). Moreover, comments preceded by '%%' will *not* be logged on the 'command log file' to avoid irrelevant logging of program information when using an unedited 'command log file' as a 'command input file'.

4.1.5 Files used by Preframe

The file environment of Preframe is illustrated in Section 4.2. The file extensions (.MOD, .JNL, etc.) are given together with file descriptions.

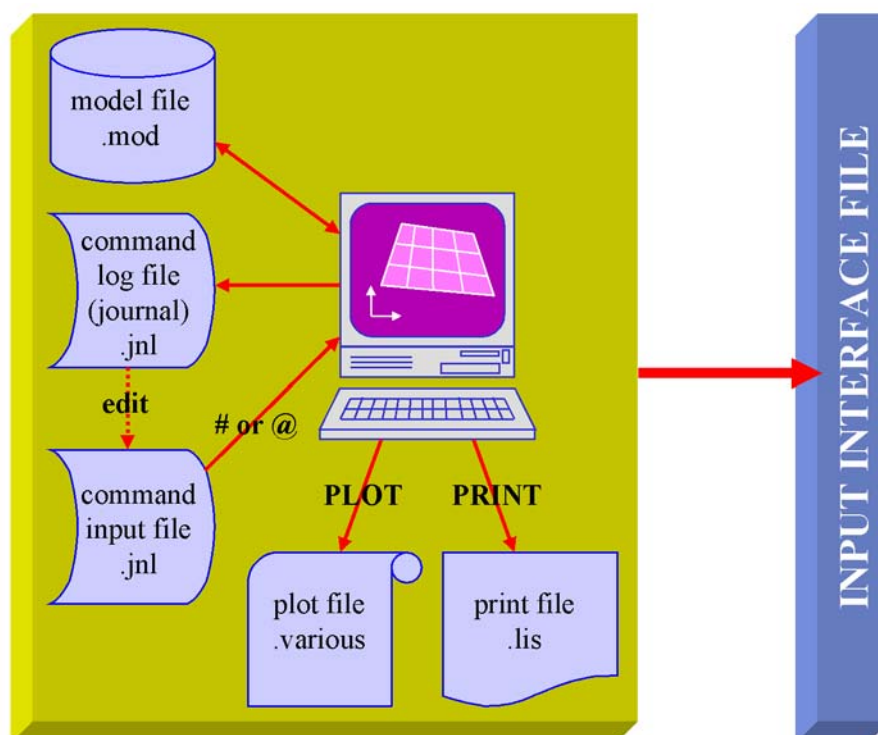


Figure 4.2 The file environment of Preframe

The files are:

- The 'command log (journal) file' (.JNL) is an ASCII file on which all commands and data given to the program are logged. This means that both data typed (or clicked) by the user and data read by the program from a 'command input file' will be logged. However, commands not changing the model (and data base), e.g. a command displaying data, will not be logged. The time of opening and closing the 'model file' is also logged. The file is very useful as a backup file both for verification purposes and for later use as a 'command input file'. The 'command log file' can be read and modified by a text editor.
- The 'command input file' (.JNL) is an ASCII file which may be read into the program. The commands contained on this file will have the same effect as if they were given by the user directly. A 'command input file' is convenient for batch execution of Preframe; see Section 4.1.7. The file is processed by using the command 'SET COMMAND-INPUT-FILE ...' followed by '# ALL' (the latter command means: read all commands found on the file). Alternatively, you may specify a 'command input file' when starting Preframe from Manager.
- The 'model file' (.MOD) is the binary data base containing all model data. The file cannot be read by a text editor.

- The ‘print file’ (.LIS) is an ASCII file which contains tables over data requested for printing by the PRINT command.
- The ‘plot file’ contains graphic information produced by the PLOT command. The file extension will depend on the plot format chosen (see the SET PLOT FORMAT command). See Section 4.1.6 for advice on using the CGM format to include plots in reports.
- The ‘Input Interface File’ (.FEM) — termed T-file for short — contains the model to be read by a subsequent hydrodynamic or structural analysis program.

Preframe has been designed to protect the user against loss of valuable data. However, accidental loss of data may occur. This may be caused by the user by for example inadvertently deleting the ‘model file’ or it may be due to an inconsistency in the data model. Such inconsistency may occur for several reasons:

- The computer goes down.
- The disk is full, the disk quota is exhausted or user privileges are inadequate.
- There is an error in the program.

If Preframe discovers an inconsistency in the data model the program will normally close all files opened and abort the execution. Preframe may then be restarted using the ‘model file’. In some cases, however, it will not be possible to resume normal execution due to an irrecoverable inconsistency.

If the ‘model file’ is lost it can be reconstructed by re-executing the program and reading input from the ‘command log file’, i.e. using it as a ‘command input file’.

Note: The ‘model file’ will normally not be compatible between different versions of Preframe. The ‘command log file’ may, however, be used as input to a new version.

4.1.6 Creating Plots for Reports

The CGM plot format (see the SET PLOT FORMAT command) is well suited for importing SESAM plots into reports produced by MS Word and other word processors. You may also transfer CGM files from one operating system to another, just make sure to use the ‘binary’ option when transferring the file with FTP (or another protocol).

Depending on the capabilities of your word processor the PostScript plot format may also be used for the purpose of importing SESAM plots into reports. Contrary to CGM, PostScript is an ASCII formatted file and is therefore more easily transferred from one computer make to another.

Note that a word processor will normally recognise only one picture (display) on each file. You should, therefore, specify a new file name for each plot command using SET PLOT FILE command.

4.1.7 Background Execution

On Unix the user may find it convenient to execute Preframe as a background job rather than as an interactive session. Here is a proposal for how to do this. This proposal is not relevant for executing Preframe through Manager in which case background execution is controlled by Manager.

Execute Preframe in the background as follows:

- Prepare a file (e.g. a revision of a previous ‘command log file’) containing the input data, let the name of the file be FILE_IN.JNL.
- Prepare a file with the following contents (the entries FILE and FILE_IN are example file names and sup-el-no is the desired superelement number):

```
' '
```

```
FILE
NEW
sup-el-no
SET COMMAND-INPUT-FILE FILE_IN
..
# ALL
..
EXIT
```

The two apostrophes in the first line enclose a blank space (it may also be a blank line) to specify a void prefix. If a prefix is given, e.g. PREFIX, it will precede the given command input file name requiring the full name of the file containing the input data to be PREFIXFILE_IN.JNL.

- Start Preframe as a background job with the file above as input file.

4.1.8 Command Line Arguments

It is possible to specify command line arguments when starting Preframe. A command line argument will influence the program execution in various ways. On Unix systems the command line arguments are simply added to the command for starting the program:

```
prompt> preframe /NOHEADER/STAT=OLD/INT=LINE/C-F=TEST_IN.JNL/FORCED-EXIT
```

The command line arguments are:

/PREFIX=text	General file name prefix
/NAME=text	General file name
/STATUS=text	Data base / journal file status
/INTERFACE=LINE	Start the program in line-mode.
/INTERFACE=PICK	Start the program in graphical user interface mode.
/HEADER=NONE	Do not show the program header.
/NOHEADER	Do not show the program header.
/HEADER=SHORT	Show the standard program header.
/WRITE-SUPERELEMENT=number	Write an Input Interface File with the given superelement number when exiting the program.
/NOWRITE-SUPERELEMENT	Do not write an Input Interface File.

<code>/COMMAND-FILE=filename</code>	Read the specified command input file after the model/journal file has been accepted.
<code>/NOCOMMAND-FILE</code>	Do not read a command input file.
<code>/FORCED-EXIT</code>	Force EXIT after initialisation and after processing of the file defined by the <code>/COMMAND-FILE</code> argument.
<code>/NOFORCED-EXIT</code>	Disable FORCED-EXIT.
<code>/EYEDIR-X=value</code>	Set initial eye direction X-value.
<code>/EYEDIR-Y=value</code>	Set initial eye direction Y-value.
<code>/EYEDIR-Z=value</code>	Set initial eye direction Z-value.
<code>/PLOT-FORMAT=format</code>	Set the default plot format to the specified format.
<code>/PLOT-COLOUR=ON (or OFF)</code>	Switch colours for plot file on (or off).
<code>/PLOT-PAGE-SIZE=size</code>	Set the default plot page size.
<code>/PLOT-ORIENTATION=orientation</code>	Set the default plot orientation.
<code>/PRINT-FORMFEED=format</code>	Set the form-feed (page break) character to either ASCII character 12 (format=ASCII) or to the FORTRAN standard of 1 in the first column (format=FORTRAN). ASCII format is default and will give proper page breaks when printing on laser printers and when importing into word processors.
<code>/WINDOW-SIZE=value</code>	Set the size of the graphical user interface window (or graphic display window). This is available on Unix only. The value to give is percentage of screen height. By default size=90.

Note the following about how to enter command line arguments:

- Command line arguments and values can be abbreviated.
- Each argument name must begin with a slash (/) and each argument value must be preceded by an equal sign (=). Spaces can freely be distributed around the equal sign and before each slash.
- Texts with blank spaces and special characters (e.g. file names) must be enclosed in quotes. Note that some operating systems change the case of the input text if it is not enclosed in quotes.
- Slanted arguments or values indicate that these are defaults.
- If at least one of the arguments `/PREFIX`, `/NAME` and `/STATUS` is specified then the prompt for data base and journal file name is skipped and defaults are used for any unspecified values.
- The values given to the `/EYEDIR` are real values. The default is the Preframe default values. If one of the three are given the other two are set to 0.0 unless specified.

- `/PLOT-FORMAT=POSTSCRIPT` involves that Preframe plots use the POSTSCRIPT format. Legal values are those presented in the SET PLOT FORMAT command. `/PLOT-FORMAT=SESAM-NEUTRAL` is the default format.
- `/PLOT-COLOUR=ON` is only available when `/PLOT-FORMAT=POSTSCRIPT` or another format supporting colours have been specified. Default is `/PLOT-COLOUR=OFF`.
- `/PLOT-PAGE-SIZE` is ignored if the plot-format is SESAM-NEUTRAL. The default is `/PLOT-PAGE-SIZE=A4`.
- `/PLOT-ORIENTATION=PORTRAIT` is the default orientation. The argument is available only for formats supporting different orientations.

4.2 Program Requirements

4.2.1 Execution Time

The execution time required is negligible for most commands. A few commands, however, will require some CPU and should be used with care on low capacity computers. An example of such is the PROPERTY GAP command.

4.2.2 Storage Space

The initial size of the data base (prior to any modelling) is less than 1 MB. 5 MB will be sufficient for most models.

4.3 Program Limitations

Graphics Devices

The graphical user interface is implemented for OSF/Motif X Window, Windows 98, Windows NT and Windows 2000. Under OSF/Motif X Window window stretching is disallowed, use the `/WINDOW-SIZE` command line argument instead.

Memory

Preframe allocates memory buffers for access to data of the data base file. When using the graphical user interface Preframe will allocate memory for the display.

- **File access buffer**

The memory is allocated when Preframe is started and the amount is fixed until exiting the program. The amount of memory allocated can be changed by editing the configuration (password) file. To change the amount insert (or modify) the line:

`MSIZE-PREFRAME-BUFFER buffer-bytes`

where buffer-bytes represents the amount of memory Preframe will allocate in bytes. The default value is 2457600 (2.4576 millions) representing 150 buffers of 16384 bytes each. The buffer should be changed

if, for example, there is not enough memory to use the graphical user interface. Note, however, that increasing the memory for buffers will not improve performance much.

- **Memory for graphical user interface**

The graphic-mode window will use memory and allocate it when needed. Large displays will need more memory than small displays.

Typing

While typing a command using the keyboard you cannot click commands in menus or select nodes and elements by clicking or use the mouse in any other way until the Return key has been hit or until the typed text has been deleted by backspace.

5 COMMAND DESCRIPTION

The hierarchical structure of the commands and numerical data is documented in this chapter by use of tables. How to interpret these tables is explained below. Examples are used to illustrate how the command structure may diverge into multiple choices and converge to a single choice.

In the example below command A is followed by either of the commands B and C. Thereafter command D is given. Legal alternatives are, therefore, A B D and A C D.

A	B	D
	C	

In the example below command A is followed by three selections of either of commands B and C as indicated by *3. For example: A B B B, or: A B B C, or A C B C, etc.

A	B	*3
	C	

In the example below the three dots in the left-most column indicate that the command sequence is a continuation of a preceding command sequence. The single asterisk indicate that B and C may be given any number of times. Conclude this sequence by the command END. The three dots in the right-most column indicate that the command sequence is to be continued by another command sequence.

...	A	B	*	...
		C		
		END		

In the example below command A is followed by any number of repetitions of either of the sequences B D and C D. Note that a pair of braces ({ }) is used here merely to define a sequence that may be repeated. The braces are not commands themselves.

A	{	B	D	}*
		C		

The characters A, B, C and D in the examples above represent parameters being COMMANDS (written in upper case) and numbers (written in lower case). All numbers may be entered as real or integer values. Brackets ([]) are used to enclose optional parameters.

Note: The command **END** is generally used to end repetitive entering of data. Using double dot (..) rather than **END** to terminate a command will, depending on at which level in the command it is given, save or discard the data entered. Generally, if the data entered up to the double dot is complete and self-contained the double dot will save the data. If in doubt, it is always safest to leave a command by entering the required number of **END** commands.

5.1 The Node and Element Select Features

Selecting nodes and elements is relevant in many different commands, e.g. the **BOUNDARY**, **DEFINE SET**, **LOAD**, **DISPLAY** and **PRINT** commands. This may be done in different ways:

- select a single node/element by referring to the node/element number,
- select a group of nodes/elements by referring to the number of the first, the last and the step in the numbering sequence,
- select all nodes/elements on a straight line segment by referring to the two end nodes of the line,
- select all nodes/elements in a plane by referring to three nodes in the plane,
- select nodes/elements contained in a previously defined set (see the **DEFINE SET** command),
- select nodes/elements by use of rubberband select (graphical mode),
 - a Selection inside a rectangle: When ready to select nodes or elements, press left mouse button (LMB) and drag a rectangle around wanted selection. This process may be repeated or combined with other select methods to finalize the selection.
 - b Selection inside a polygon: When ready to select nodes or elements, draw the polygon line as explained below:
 - 1 Position cursor and press shift key to define the first polygon point.
 - 2 While keeping the shift key pressed, repeatedly move the cursor and click the LMB to make the polygon. (If the LMB is pressed rather than clicked, a rubberband line appears as an aid to determine the position of the polygon segment.)
 - 3 Release shift key and click LMB to define the last polygon point.
 - 4 A straight line between the first and last polygon points closes the polygon.
- use a mixture of the above selection criteria,
- select all nodes/elements in the model.

Note: Whenever a node or element is to be selected the user may either enter the node/element number using the keyboard or may — when using the graphical user interface; see Section 3.1 — click the left mouse button on the appropriate node/element. The availability of such graphical selection is indicated by an informative text in the ‘pick indication area’, see Figure 3.2.

Whenever selecting nodes is relevant the following text will appear:

SELECT NODES?

and the select options are:

nodeno				*
GROUP	node1	node2	nstep	
LINE	node1	node2		
PLANE	node1	node2	node3	
SET	setname			

ALL
NO

Whenever selecting elements is relevant the following text will appear:

SELECT ELEMENTS?

and the select options are:

elno				*
GROUP	elno1	elno2	estep	
LINE	node1	node2		
PLANE	node1	node2	node3	
BY-SECTION	secnum			
BY-MATERIAL	matum			
CONNECTED-TO-NODE	node-select			
SET	setname			

ALL
NO

Having selected some nodes/elements the program will request the next select directive by either of the prompts (unless the ALL option is given in which case the select mode is terminated):

SELECT MORE NODES? /NO/
SELECT MORE ELEMENTS? /NO/

‘NO’ is the default answer and the select mode is terminated by hitting Return or by entering ‘NO’. If more nodes/elements are to be selected, one of the select options described above are given. Several select options can be given on the same input line.

Note that you should not terminate the select mode by ‘.’ as this will abort the current command and no nodes/elements will be selected.

PARAMETERS:

nodeno	Number of a node to be selected.
elno	Number of an element to be selected.
GROUP	A group of nodes or elements are to be selected.
LINE	Select the nodes or the elements positioned on a straight line segment defined by two nodes. The tolerance or 'thickness' of the line is defined by the SET COORDINATE-TOLERANCE command.
PLANE	Select the nodes or the elements positioned in a plane defined by three nodes. The tolerance or 'thickness' of the plane is defined by the SET COORDINATE-TOLERANCE command.
BY-SECTION	Select elements with specific section property.
BY-MATERIAL	Select elements with specific material property.
CONNECTED-TO-NODE	Select the elements connected to selected nodes.
SET	Select the nodes or the elements contained in a previously defined set.
node <i>i</i>	Node numbers defining a GROUP, LINE or PLANE, <i>i</i> is 1, 2, ...
nstep	The step in the node numbering defining a GROUP of nodes to be selected.
elno <i>i</i>	Element numbers defining a GROUP of elements, <i>i</i> is 1, 2, ...
estep	The step in the element numbering defining a GROUP of elements to be selected.
secnum	The number of a property section.
matnum	The number of a property material.
matnum	The number of a property material.
node-select	Select nodes by node select options.
ALL	Select all nodes or elements in the structure.
NO	Terminate the select mode.

5.2 Detailed Description of Commands

The input commands are described in the following. The commands and sub-commands are described in alphabetic order. Below is a list of all main (basic level) commands.

ADD-DISPLAY
ALIGN
ASSIGN
BOUNDARY
CHANGE
COPY
CREATE
DEFINE
DELETE
DISPLAY
ELEMENT
EXIT
GENERATE
HELP
INITIAL-CONDITION
LABEL
LINEAR-DEPENDENCY
LOAD
MASS-ON-NODE
NODE
PLOT
PRINT
PROPERTY
RE-DISPLAY
READ
RENUMBER
ROTATE
SET
SPLIT
TRANSFORMATION
WRITE
ZOOM
#

ADD-DISPLAY

ADD-DISPLAY	LOAD	load-case	load-type
	SOIL-PROFILE	soil-id	

PURPOSE:

The command adds display of loads for a selected load case or adds display of soil-profile.

PARAMETERS:

LOAD	<p>Add display of a selected load case.</p> <p>If arrow presentation is selected, the load is presented as arrows with their heads where the load applies. The arrow lengths are proportional with the magnitude of the load. The largest of the arrows will have a length on the display of approximately 15 mm, this length may be adjusted by the SET GRAPHICS SIZE-SYMBOLS LOAD-ARROWS command. The tails of the arrows are connected by dotted lines (relevant for element distributed loads only).</p> <p>Only the part of the load related to the currently displayed elements will be added. The positions of the arrows are adjusted for shrunk elements and beam eccentricities.</p>
load-case	Load case to add. If a load case contains several load types (node force, element distributed, etc.) then only one of the load types may be added at a time.
load-type	Type of load to add. See the LOAD command for the different types of loads.
SOIL-PROFILE	Add to display nodes and elements representing the soil profile.
soil-id	The soil profile id (number) to be used. Currently only id 1 allowed.

NOTES:

The SET GRAPHICS PRESENTATION LOAD command controls how to present the load.

Use the ADD LOAD command prior e.g. zooming in or adding element or node numbers.

Nodes and elements generated to represent the soil profile will be assigned node and element numbers starting with 800000.

The command may be activated (for soil id 1) by the button 'Soil On' under Display: in the graphical user interface quick button area.

The soil profile will automatically be delete prior to writing the SESAM Interface File (.FEM).

The soil profile will automatically deleted after modification of the soil profile data.

On the display, the nodes defining the soil division (types) are drawn in blue (equal to supernodes) while nodes defining internal layers are drawn in yellow (equal to ordinary nodes).

The soil profile is by default 'drawn' at global co-ordinates $X=0.0$, $Y=0.0$. This may however be changed by the command SET SOIL-PROFILE-X-Y.

See also:

```
SET SOIL-PROFILE-X-Y...  
SET GRAPHICS PRESENTATION LOAD...
```

EXAMPLES:

```
ADD-DISPLAY LOAD 10 NODE-FORCE  
ADD-DISPLAY SOIL-PROFILE 1
```

ALIGN

ALIGN	elem1	ALIGNED-WITH	elem2
	LINE	node1	node2

PURPOSE:

The command specifies that elements shall be aligned so that changes to the position of either of the two extreme nodes will result in the common node moving to a new position to maintain the alignment (or establish alignment if not aligned prior to giving the command). See Section 3.4.

PARAMETERS:

elem1	Align elements elem1 and elem2.
elem2	
LINE	Add alignment attribute to all elements on the straight line between node1 and node2.
node1	Start node for alignment.
node2	End node for alignment.

NOTES:

If the node in one end of aligned elements moves, all intermediate nodes will be moved to keep the elements in a straight line. Also, if the first or last element is updated with eccentricity information, all intermediate elements will be applied with necessary eccentricities to keep elements in a straight line.

The centre node in an X-brace will move when gap calculations (adding brace eccentricities) are performed at the structural joints. Hence, if eccentricities are deleted by the command DELETE ECCENTRICITY and the elements selected are part of an X-brace then the elements will no longer be on a straight line. To align the elements use the command CHANGE NODE node ; where node is a node at the end of any chain of aligned elements and the ; (semicolon) indicates that existing X, Y and Z co-ordinates shall be used. This command will actually align all chains of aligned elements independent of location in the model.

See also:

SET ALIGNMENT-AUTOMATIC ...

ASSIGN

ASSIGN	CAN	sub-commands
	CONE	
	HYDRODYNAMIC	
	PILE-DATA	
	SEGMENT	
	SOIL-DATA	
	STABILITY	
	STUB	

PURPOSE:

The command is used to modify structural joints and to add conceptual attributes.

PARAMETERS:

CAN	Add joint chord strengthening.
CONE	Add a conical transition to a member
HYDRODYNAMIC	Add hydrodynamic attributes to member.
PILE-DATA	Add pile attributes to piles.
SEGMENT	Add a new segment to member.
SOIL-DATA	Add soil data to the soil profile.
STABILITY	Add stability attributes to member.
STUB	Add joint brace strengthening.

The sub-commands and data are fully explained on the following pages.

ASSIGN CAN

...	CAN	CHORD	node	element	dy	thk	sfy	sfz	length	...
		JOINT	node							

or if the SET ASSIGN-OPTION MANUAL-NUMBERING is switched ON:

...	CAN	CHORD	node	element	nodeno	eleno	dy	thk	sfy	sfz	length	...
		JOINT	node									

or if the SET ASSIGN-OPTION SECTION-NUMBER is switched ON:

...	CAN	CHORD	node	element	secno	length	...
		JOINT	node				

or if both above alternatives are switched ON:

...	CAN	CHORD	node	element	nodeno	eleno	secno	length	...
		JOINT	node						

and if the modified element is not part of an existing member concept:

...	name	node1	node2
-----	------	-------	-------

PURPOSE:

The command assigns a can section to member end, to one of the incoming chords (chord + aligned) or to both chords entering the joint. See Section 3.6.1 regarding the ASSIGN-OPTION switches.

PARAMETERS:

CHORD	Assign to selected part of joint, chord or aligned chord.
JOINT	Assign to both chord and aligned chord in joint.
node	Node for start of can section.
element	Element to modify.
dy	Pipe outer diameter (default = element to split).
thk	Thickness of pipe wall (default = element to split).
sfy	Pipe section shear area modifying factor, local y axis.
sfz	Pipe section shear area modifying factor, local z axis.
length	Can length.

nodeno	Number of the node to create. AUTO may also be given.
eleno	Number of the element to create. AUTO may also be given.
secno	Section number to be used as can strengthening.
name	Name of member to be defined.
node1	Existing node defining start of member.
node2	Existing node defining end of member.

NOTES:

For the JOINT option, the pipe section parameters must be given twice (chord + aligned). The default can length is calculated according to given parameters (see command SET CAN-STUB-LENGTH-PARAMETERS) and joint geometry.

If the pipe section parameters given do not correspond to an existing pipe section, a new section will automatically be created.

Elements belonging to a member may at any time be modified with respect to section and material by use of the command PROPERTY CONNECT.

See also:

```
CHANGE CAN ...
SET CAN-STUB-LENGTH-PARAMETERS ...
```

EXAMPLES

```
ASSIGN CAN JOINT 5 ;
```

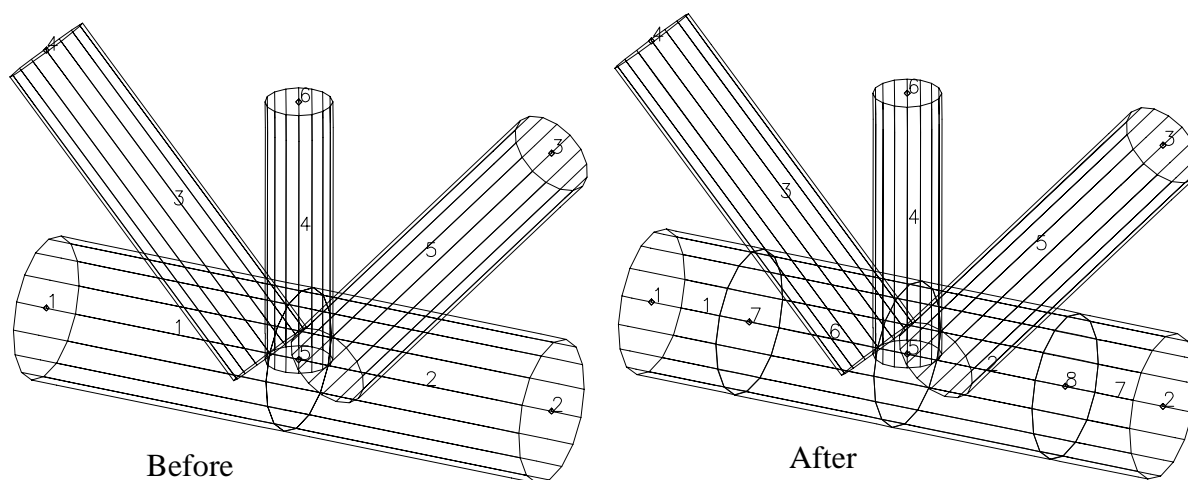


Figure 5.1 Create can sections by ASSIGN CAN

ASSIGN CONE

...	CONE	node	element	ANGLE	angle	dy	thk	sfy	sfz	...
				LENGTH	length					

or if the SET ASSIGN-OPTION MANUAL-NUMBERING is switched ON:

...	CONE	node	element	nodeno	eleno	ANGLE	angle	dy	thk	sfy	sfz	...
						LENGTH	length					

or if the SET ASSIGN-OPTION SECTION-NUMBER is switched ON:

...	CONE	node	element	ANGLE	angle	secno	...
				LENGTH	length		

or if both above alternatives are switched ON:

...	CONE	node	element	nodeno	eleno	ANGLE	angle	secno	...
						LENGTH	length		

and if the modified element is not part of an existing member concept:

...	name	node1	node2
-----	------	-------	-------

PURPOSE:

The command assigns a cone section to a member. The conical transition will in PREFRAME (and in the analysis) be represented by one element without tapered section. The reference node given shall have two incoming elements with pipe section of different outer diameter. See Section 3.6.1 regarding the ASSIGN-OPTION switches.

PARAMETERS:

node	Reference node for start of cone.
element	Element to modify.
ANGLE	Use the angle option to define cone length.
LENGTH	Use the length option to define cone length.
angle	Cone angle (default corresponding to 1 : 6).
length	Cone length (default corresponding to angle 1 : 6).
dy	Pipe outer diameter (default = average of the two elements entering the ref. node).
thk	Thickness of pipe wall (default = largest of the two elements entering ref. node).

sfy	Pipe section shear area modifying factor, local y axis.
sfz	Pipe section shear area modifying factor, local z axis.
nodeno	Number of the node to create. AUTO may also be given.
eleno	Number of the element to create. AUTO may also be given.
secno	Section number to be used as cone element.
name	Name of member to be defined.
node1	Existing node defining start of member.
node2	Existing node defining end of member.

NOTES:

If the pipe section parameters given do not correspond to an existing pipe section, a new section will automatically be created.

Elements belonging to a member may at any time be modified with respect to section and material by use of the command PROPERTY CONNECT.

See also:

CHANGE CONE ...
PROPERTY CONNECT ...

ASSIGN HYDRODYNAMIC

...	HYDRODYNAMIC	DRAG-COEFFICIENT	cdx	cdy	cdz	...
		FLOODING-COEFFICIENT	floo			
		INERTIA-COEFFICIENTS	cmx	cmy	cmz	

...	ALL
	BY-ELEMENT
	BY-NAME

PURPOSE:

The command assigns hydrodynamic coefficients to members.

PARAMETERS:

DRAG-COEFFICIENT	Assign drag coefficients.
cdx	Drag coefficient value, local x-axis.
cdy	Drag coefficient value, local y-axis.
cdz	Drag coefficient value, local z-axis.
FLOODING-COEFFICIENT	Assign flooding status.
floo	Flooding parameter (0.0 = not flooded, 1.0 = completely flooded).
INERTIA-COEFFICIENT	Assign inertia coefficients.
cmx	Inertia coefficient value, local x-axis.
cmy	Inertia coefficient value, local y-axis.
cmz	Inertia coefficient value, local z-axis.
ALL	Assign to all existing members.
BY-ELEMENT	Assign to existing member containing at least one of the selected elements. (Select elements by use of standard select element options.) If the selection contains elements that are not part of any members, new members will be created with the member name equal to element number.
BY-NAME	Assign to members according to specified names (NO to end name list).

NOTES:

Assigned hydrodynamic values may be labelled by use of the command LABEL CONCEPT-ATTRIBUTES HYDRODYNAMIC.

Hydrodynamic coefficients will be changed / overwritten by a new ASSIGN command.

See also:

`LABEL CONCEPT-ATTRIBUTES ...`

ASSIGN PILE-DATA

...	PILE-DATA	FIXED-TO	node	... select-option			
		DENSITY-FLUID	dens	... concept-select			
		STIFFNESS-MODIFIER	ea	ei	ga	gip	... concept-select
		TIP-CODE	tip	... select-option			
		YIELD-STRENGTH	yield	... concept-select			

where the concept-select alternatives are:

...	ASSEMBLY	... select-option
	CONCEPT	
	ELEMENT	

where the select-option alternatives are:

...	ALL
	BY-ELEMENT
	BY-NAME

PURPOSE:

The command assigns concept attributes to the selected pile concepts or part of concept.

PARAMETERS:

FIXED-TO	Assign pile group fixed to node reference.
DENSITY-FLUID	Assign unit density of fluid / soil inside the pile.
STIFFNESS-MODIFIER	Assign element stiffness modifier data.
TIP-CODE	Assign pile tip boundary condition code.
YIELD-STRENGTH	Assign pile material yield strength.
ASSEMBLY	Assign attribute to complete assembly (currently not in use).
CONCEPT	Assign attribute to whole pile concept (parent concept).
ELEMENT	Assign attribute to selected pile elements (part concepts, currently not in use due to limitations in Splice).
ALL	Assign to all existing piles.

BY-ELEMENT	Assign to existing piles containing at least one of the selected elements. (Select elements by use of standard select element options.)
BY-NAME	Assign to piles according to specified names (NO to end name list).
node	Node in structure to rigidly connect the pile head (used for pile groups).
dens	Value of the density-fluid attribute.
ea	Axial stiffness value to replace current one (if not zero).
ei	Bending stiffness value to replace current one (if not zero).
ga	Shear stiffness value to replace current one (if not zero).
gip	Torsion stiffness value to replace current one (if not zero).
tip	Value of the tip-code attribute.
yield	Value of the yield strength attribute.

NOTES:

The concept attribute is only connected to a pile concept if relevant, e.g. FIXED-TO information will only be connected to a pile concept defined as part of a pile group.

FIXED-TO and TIP-CODE attributes may only be assigned to parent concept.

Allowable TIP-CODE data are:

0 = Pile tip is free

1 = Pile tip is fixed

2 = The pile is assumed to be infinitely long beneath the tip

3 = Identical to code 2, except that the axial stiffness is replaced by a secant value computed from pile tip q-z data given as input to the soil data

-ND = ND is a nodal point number (internal pile numbering with 1 at pile head). The pile will be assumed infinitely long with respect to lateral and torsional solution below node ND.

To users familiar with PILGEN: Note that the DENSITY-FLUID attribute corresponds to the gamma-fluid parameter previously given in the PILGEN.INP file, but it is now the unit density, and not the unit weight, that shall be given.

For pile groups, the 'fixed-to' node attribute (i.e. the node to which all piles in the pile group shall be rigidly connected) must be assigned to the piles. The 'fixed-to' node reference used for pile group must have equal Z-coordinate as the pile heads.

See also:

LABEL CONCEPT-ATTRIBUTES . . .

ASSIGN SEGMENT

...	SEGMENT	node	element	length	...
-----	---------	------	---------	--------	-----

or if the SET ASSIGN-OPTION MANUAL-NUMBERING is switched ON:

...	SEGMENT	node	element	nodeno	eleno	length	...
-----	---------	------	---------	--------	-------	--------	-----

and if the modified element is not part of an existing member concept:

...	name	node1	node2
-----	------	-------	-------

PURPOSE:

The command assigns a new segment to a member. The new element (segment) will inherit section properties from the original element.

PARAMETERS:

node	Node for start of segment section.
element	Element to modify.
length	Segment length.
nodeno	Number of the node to create. AUTO may also be given.
eleno	Number of the element to create. AUTO may also be given.
name	Name of member to be defined.
node1	Existing node defining start of member.
node2	Existing node defining end of member.

NOTES:

Elements belonging to a member may at any time be modified with respect to section and material by use of the command PROPERTY CONNECT.

The switch SET ASSIGN-OPTION MANUAL-NUMBERING can be used in connection with the ASSIGN SEGMENT command. When switched ON the user may manually give node and element number to the node and element created by this command.

See also:

PROPERTY CONNECT ...

SET ASSIGN-OPTION MANUAL-NUMBERING ...

ASSIGN SOIL-DATA

...	SOIL-DATA	PY-TZ-QZ-CODE	...	NODE	node	... data-set ...
		SKIN-FRICTION		Z-LEVEL	z-level	
		TIP-RESISTANCE				

where the data-set for PY-TZ-QZ-CODE is defined through:

...	py-code	tz-code	qz-code
-----	---------	---------	---------

where the data-set for SKIN-FRICTION is defined through:

...	skin-cmp	skin-tns	g0-soil	ds/dia-rat	pois	dt/dia-rat
-----	----------	----------	---------	------------	------	------------

where the data-set for TIP-RESISTANCE is defined through:

...	skin-cmp	skin-tns	g0-soil	ds/dia-rat	sig-tip	pois	dt/dia-rat
-----	----------	----------	---------	------------	---------	------	------------

PURPOSE:

The command is used to define and assign the PY-TZ-QZ codes, skin friction parameters and tip resistance parameters to the soil profile. The data are used when creating the GENSOD input file. The Z level may be given by node selection or be manual input.

PARAMETERS:

PY-TZ-QZ-CODE	Define PY-TZ-QZ codes.
SKIN-FRICTION	Define skin friction parameters.
TIP RESISTANCE	Define tip resistance parameters.
NODE	Define the Z level by selecting a node.
Z-LEVEL	Define the Z level by manually giving the z-value.
node	The node defining the Z level where values are given.
z-level	The Z level where values are given.
py-code	PY code to be used.
tz-code	TZ code to be used.
qz-code	QZ code to be used.
skin-cmp	Peak skin friction in compression.
skin-tns	Peak skin friction in tension.

g0-soil	Initial value of soil shear modulus.
ds/dia-rat	Ratio between displacement to reach peak skin friction and pile diameter.
sig-tip	Peak tip stress acting at pile tip.
pois	Soil Poisson ratio.
dt/dia-rat	Ratio between displacement to reach peak tip stress and pile diameter.

NOTES:

The data-sets defined will be active from the given Z level reference and down to next Z level where similar data-set is defined.

When using the NODE option to define the Z level, the command will be logged onto the journal file as 'Z-LEVEL -z.zz', where -z.zz equals the z co-ordinate defined by the selected node.

This command may also be used to modify existing data.

See also the Gensod User Manual for specific soil related explanation.

ASSIGN STABILITY

...	STABILITY	BUCKLING-FACTOR	MANUAL	ky	kz	...
			NPD-NS-3472			
		BUCKLING-LENGTH	LENGTH-BETWEEN-JOINTS			
			MANUAL	ly	lz	

...	ALL
	BY-ELEMENT
	BY-NAME

PURPOSE:

The command assigns stability buckling parameters to members.

PARAMETERS:

BUCKLING-FACTOR	Assign buckling factors.
MANUAL	Use manually given buckling factors.
NPD-NS-3472	Use buckling factors according to NPD / NS-3472.
ky	Buckling factor (effective length factor), local y-axis.
kz	Buckling factor (effective length factor), local z-axis.
BUCKLING-LENGTH	Assign buckling length.
LENGTH-BETWEEN-JOINTS	Use buckling length = length between joints.
ly	Buckling length, local y-axis.
lz	Buckling length, local z-axis.
ALL	Assign to all existing members.
BY-ELEMENTS	Assign to existing member containing at least one of the selected elements. (Select elements by use of standard select element options.) If the selection contains elements that are not part of any members, new members will be created with the member name equal to element number.
BY-NAME	Assign to members according to specified names (NO to end name list).

NOTES:

Assigned stability parameters may be labelled by use of the command LABEL CONCEPT-ATTRIBUTES STABILITY.

Stability parameters will be changed / overwritten by a new ASSIGN command.

See also:

LABEL CONCEPT-ATTRIBUTES ...

ASSIGN STUB

...	STUB	BRACE	node	element	dy	thk	sfy	sfz	length	...
		JOINT	node							

or if the SET ASSIGN-OPTION MANUAL-NUMBERING is switched ON:

...	STUB	BRACE	node	element	nodeno	eleno	dy	thk	sfy	sfz	length	...
		JOINT	node									

or if the SET ASSIGN-OPTION SECTION-NUMBER is switched ON:

...	STUB	BRACE	node	element	secno	length	...
		JOINT	node				

or if both above alternatives are switched ON:

...	STUB	BRACE	node	element	nodeno	eleno	secno	length	...
		JOINT	node						

and if the modified element is not part of an existing member concept:

...	name	node1	node2
-----	------	-------	-------

PURPOSE:

The command assigns a stub section to a member end, to one of the incoming braces, or to all braces entering the joint. See Section 3.6.1 regarding the ASSIGN-OPTION switches.

PARAMETERS:

BRACE	Assign to selected brace.
JOINT	Assign to all braces in joint.
node	Node for start of stub section.
element	Element to modify.
dy	Pipe outer diameter (default = element to split).
thk	Thickness of pipe wall (default = element to split).
sfy	Pipe section shear area modifying factor, local y axis.
sfz	Pipe section shear area modifying factor, local z axis.
length	Stub length.

nodeno	Number of the node to create. AUTO may also be given.
eleno	Number of the element to create. AUTO may also be given.
secno	Section number to be used as stub strengthening.
name	Name of member to be defined.
node1	Existing node defining start of member.
node2	Existing node defining end of member.

NOTES:

For the JOINT option, the pipe section parameters must be given for each brace. The default stub length is calculated according to given parameters (see command SET CAN-STUB-LENGTH-PARAMETERS) and joint geometry.

If the pipe section parameters given do not correspond to an existing pipe section, a new section will automatically be created.

Elements belonging to a member may at any time be modified with respect to section and material by use of the command PROPERTY CONNECT.

See also:

```
CHANGE STUB ...
SET CAN-STUB-LENGTH-PARAMETERS ...
PROPERTY CONNECT ...
```

EXAMPLES

```
ASSIGN STUB JOINT 5 ;
```

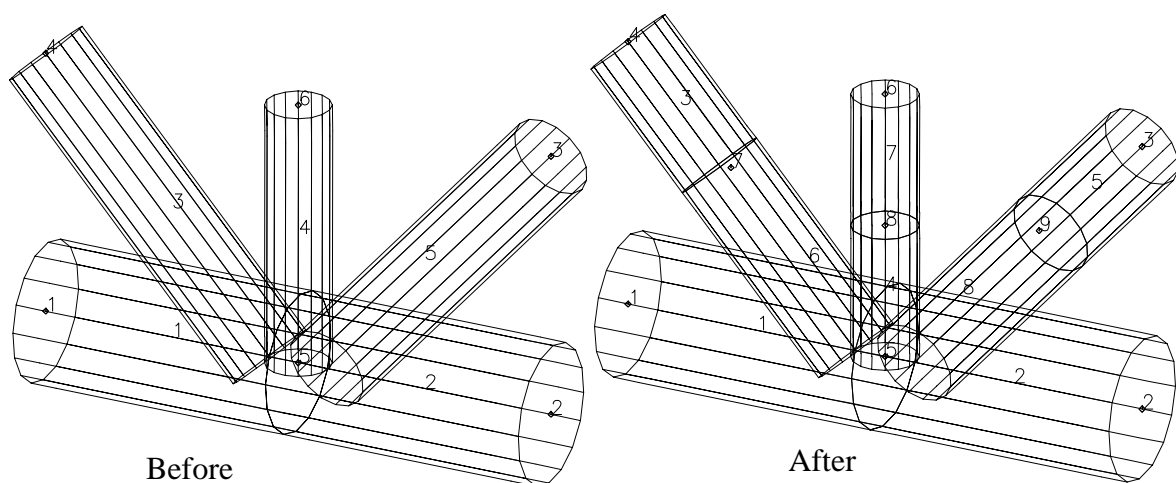


Figure 5.2 Create stub sections by ASSIGN STUB

BOUNDARY

BOUNDARY	FREE	*6	...	GLOBAL		select-nodes
	FIXED					
	PRESCRIBED			TRANSFORMATION		
	SUPER					
	fixcode					

PURPOSE:

The command defines boundary conditions for the degrees of freedom (d.o.f.s) of the nodes.

PARAMETERS:

FREE	Free to move
FIXED	Fixed at zero displacement
PRESCRIBED	With a prescribed displacement or acceleration (to be given by the LOAD command)
SUPER	Super d.o.f.
fixcode	Fixation code (integer), 0 = FREE, 1 = FIXED, 2 = PRESCRIBED and 4 = SUPER
GLOBAL	Boundary conditions are specified in the global coordinate system.
TRANSFORMATION	Boundary conditions are specified in a rotated coordinate system.
trano	Transformation reference number.
select-nodes	Select nodes; see Section 5.1.

NOTES:

In addition, the boundary conditions LINEAR and SUPERL are defined using the LINEAR-DEPENDENCY command. SUPERL has exactly the same effect as SUPER, only that it was defined within the LINEAR-DEPENDENCY command.

A node for which no boundary conditions are defined is FREE for all d.o.f.s. A boundary condition code is given for all six d.o.f.s individually. Even for nodes with only 3 d.o.f.s, six codes must be given.

Boundary conditions can be given in a rotated coordinate system by the TRANSFORMATION option.

A nodal load may be given for any boundary condition. The PRESCRIBED boundary condition is for defining a prescribed displacement or acceleration (LOAD lc NODE PRESCRIBED-DISPLACEMENT ...).

See also:

```
DELETE BOUNDARY ...  
LABEL BOUNDARY-CONDITION-SYMBOLS ...  
PRINT NODE BOUNDARY-CONDITIONS ...  
SET PRINT TABLE NODE-BOUNDARY-TABLE
```

CHANGE

CHANGE	CAN	sub-commands
	CONE	
	ECCENTRICITY	
	ELEMENT	
	HINGE	
	INITIAL-CONDITION	
	JOINT	
	LINEAR-DEPENDENCY	
	LOAD	
	MASS-ON-NODE	
	MATERIAL	
	NODE	
	SECTION	
	SET	
	STUB	
	TRANSFORMATION	

PURPOSE:

The command changes data previously defined. Most of the sub-commands have identical or very similar syntax with the commands defining the data. Therefore, rather than describing these sub-commands in detail here, reference is made to the commands defining the data.

However, some CHANGE commands demand special explanation which is found on the following pages

NOTES:

The CHANGE ELEMENT command changes the node(s) to which an element is connected. Element type and data assigned to the element cannot be changed. The command will, therefore, only request the new node number(s) of the element.

The CHANGE MATERIAL command changes previously defined materials. Only the parameters (the values describing the material) can be changed; the material type itself cannot be changed. Thus, it is not possible to change e.g. an AXIAL-SPRING material to a SPRING-TO-GROUND material. With the exception that the material type is not requested, the command is equal to the command for defining materials, PROPERTY MATERIAL; see this.

The CHANGE SECTION command changes previously defined cross sections. Only the parameters (the values describing the cross section) can be changed; the cross section type itself cannot be changed. Thus, it

is not possible to change e.g. a PIPE cross section to a CHANNEL cross section. With the exception that the cross section type is not requested, the command is equal to the command for defining materials, PROPERTY SECTION; see this.

CHANGE CAN

...	CAN	CHORD	node	element	dy	thk	sfy	sfz	length
		JOINT	node						

or if the SET ASSIGN-OPTION SECTION-NUMBER is switched ON:

...	CAN	CHORD	node	element	secno	length
		JOINT	node			

PURPOSE:

The command changes a can section, either one of the incoming chords (chord + aligned) or to both chords entering the joint. See Section 3.6.1 regarding the ASSIGN-OPTION switch.

PARAMETERS:

CHORD	Assign to selected part of joint, chord or aligned chord.
JOINT	Assign to both chord and aligned chord in joint.
node	Node for start of can section.
element	Element to modify.
dy	Pipe outer diameter (default = element to split).
thk	Thickness of pipe wall (default = element to split).
sfy	Pipe section shear area modifying factor, local y axis.
sfz	Pipe section shear area modifying factor, local z axis.
length	Can length.
secno	Section number to be used as can strengthening.

NOTES:

For the JOINT option, the pipe section parameters must be given twice (chord + aligned). The default can length is calculated according to given parameters (see command SET CAN-STUB-LENGTH-PARAMETERS) and joint geometry.

If the pipe section parameters given do not correspond to an existing pipe section, a new section will automatically be created.

Elements belonging to a member may at any time be modified with respect to section and material by use of the command PROPERTY CONNECT.

See also:

ASSIGN CAN . . .

SET CAN-STUB-LENGTH-PARAMETERS . . .

CHANGE CONE

...	CONE	node	element	length	dy	thk	sfy	sfz
-----	------	------	---------	--------	----	-----	-----	-----

or if the SET ASSIGN-OPTION SECTION-NUMBER is switched ON:

...	CONE	node	element	length	secno
-----	------	------	---------	--------	-------

PURPOSE:

The command changes a cone section to a member. See Section 3.6.1 regarding the ASSIGN-OPTION switches.

PARAMETERS:

node	Reference node for start of cone.
element	Element to modify.
length	Cone length.
dy	Pipe outer diameter (default = average of the two elements entering the ref. node).
thk	Thickness of pipe wall (default = largest of the two elements entering ref. node).
sfy	Pipe section shear area modifying factor, local y axis.
sfz	Pipe section shear area modifying factor, local z axis.
length	Length of cone element.
secno	Section number to be used as cone element.

NOTES:

When changing the length of a cone, the node given for start of cone will be unchanged, while the node at the opposite end of the cone element will move.

If the pipe section parameters given do not correspond to an existing pipe section, a new section will automatically be created.

Elements belonging to a member may at any time be modified with respect to section and material by use of the command PROPERTY CONNECT.

See also:

```
ASSIGN CONE ...  
PROPERTY CONNECT ...
```

CHANGE JOINT

...	JOINT	node-select	CAN-STUB-LENGTH	
			GAP-PLANEWISE	gap

PURPOSE:

The command updates joints regarding required length of can and stub sections, and for calculation of brace eccentricities to satisfy minimum gap between braces.

PARAMETERS:

node-select	Select nodes by use of ordinary select node options.
CAN-STUB-LENGTH	Adjust lengths of cans and stubs.
GAP-PLANEWISE	Calculate necessary eccentricities to adjust for gap between braces.
gap	Minimum gap between braces.

NOTES:

The can and stub lengths are updated according to the geometric rules defined by the SET CAN-STUB-LENGTH-PARAMETERS command and any changes to can / stub section geometry or updated brace eccentricity data.

When a cone segment is connected to a can / stub, the length of the cone segment is kept unchanged when the length of can / stub is adjusted.

The gaps / eccentricities are calculated for braces located within each brace plane entering the joint, hence for multi-planar / arbitrary joint layout, the PROPERTY GAP command should be used.

For braces with larger existing gap than minimum gap, the gap will not be reduced to the given gap value.

For examples of the resulting element eccentricities after use of the CHANGE JOINT node-select GAP-PLANEWISE see Appendix A 3.

See also:

SET CAN-STUB-LENGTH-PARAMETERS . . .

CHANGE LINEAR-DEPENDENCY

...	LINEAR-DEPENDENCY	dep-node	dep-dof	indep-node	indep-dof	beta	*	*
-----	-------------------	----------	---------	------------	-----------	------	---	---

PURPOSE:

The command changes linear dependencies between nodes. Only the linear dependency factor beta may be changed. The linear dependency may originally have been defined by either the GENERAL-NODE-DEPENDENCY or the TWO-NODE-DEPENDENCY option; see the LINEAR-DEPENDENCY command.

PARAMETERS:

dep-node	Node number previously defined as dependent.
dep-dof	D.o.f. previously defined as dependent of the indep-dof of the indep-node, choose either: X — Translation in x-direction Y — Translation in y-direction Z — Translation in z-direction R-X — Rotation about the x-direction R-Y — Rotation about the y-direction R-Z — Rotation about the z-direction
indep-node	Node number previously defined as independent.
indep-dof	D.o.f. previously defined as independent, choose either X — Translation in x-direction Y — Translation in y-direction Z — Translation in z-direction R-X — Rotation about the x-direction R-Y — Rotation about the y-direction R-Z — Rotation about the z-direction
beta	New linear dependency factor.

NOTES:

See also:

```
DELETE LINEAR-DEPENDENCY ...  
LINEAR-DEPENDENCY ...  
PRINT NODE LINEAR-DEPENDENCY ...
```

CHANGE LOAD

...	LOAD	load-case	ELEMENT	CONSTANT-TEMPERATURE-ACROSS-THICKNESS		...	
				DISTRIBUTED			
				LINE-LOAD			
				POINT			
			GRAVITY				
			NODE	FORCE			
				PRESCRIBED-ACCELERATION			
				PRESCRIBED-DISPLACEMENT			
			ROTATION-OF-STRUCTURE				
			TO-MASSSES				

PURPOSE:

The command changes loads previously defined by the LOAD command. Only load components previously defined may be changed, i.e. a load originally defined as an ELEMENT DISTRIBUTED load cannot be changed to an ELEMENT POINT load, or to a NODE load.

Changing ELEMENT, GRAVITY, NODE and ROTATION-OF-STRUCTURE loads is done in the same way as defining them by the LOAD command, with one exception only: a load index has to be given; see below. Rather than describing these change alternatives here reference is made to the LOAD command.

The CHANGE ... TO-MASSSES command, however, does not have its counterpart within the LOAD command and is therefore described in more detail in the following.

In changing loads, the user specifies the load case, the nodes/elements and the load index of the load to be changed. The load index is used to distinguish between different loads of the same type for the same node/element for the same load case. For example, a nodal force defined for the second time for the same node for the same load case is given index 2.

When changing the load for a single node/element the original values will be used as default values. When changing the load for several nodes/elements Preframe will propose default values corresponding to the load for the node/element with the smallest internal element number. The loads for all selected nodes/elements are then changed according to the new load values. Complex loads may be changed to real loads and vice versa.

CHANGE LOAD load-case TO-MASSSES

...	load-case	TO-MASSSES	gravity-load-case		select-nodes
			NODE-LOAD	gravity-load-case	
			ELEMENT-DISTRIBUTED-LOAD		select-elements
			ELEMENT-POINT-LOAD		

PURPOSE:

The command changes static (real) nodal forces and element loads to nodal masses. This is done by dividing the previously given force(s) of load case load-case by the acceleration of gravity, previously defined as part of load case gravity-load-case. This mass is added to any previously defined masses for the translational components of the mass matrix, mt_x , mt_y , mt_z ; the rotational components and off-diagonal terms will not be added any values (any previously defined rotational mass component will be maintained):

$$mt_x = mt_y = mt_z = f / g$$

$$mr_x = mr_y = mr_z = 0$$

PARAMETERS:

NODE-LOAD	Convert node loads, equal to selecting gravity-load-case directly after TO-MASSSES selection.
ELEMENT-DISTRIBUTED-LOAD	Convert element distributed loads.
ELEMENT-POINT-LOAD	Convert element point loads.
load-case	Load case number.
gravity-load-case	Load case for which the acceleration of gravity is used.
select-nodes	Select nodes; see Section 5.1.
select-elements	Select elements; see Section 5.1.

NOTES:

This change of load to mass is only made on the condition that the forces to change and the referred acceleration of gravity are parallel and with the same sign (the x-, y- and z-components are considered in combination and not individually). A load including moments is not changed, even if its translational part (the force) is parallel with the acceleration of gravity. Further, a complex load (a load including an imaginary part) will not be changed (this may first be changed to a real load by removing the imaginary part and then be changed to mass).

All forces (all indexes) for the selected node / element and load case load-case are changed and that part of the load case is deleted; any other type of load for load case load-case will not be affected. In other words,

the presence other types of load for the same load case do not prevent the nodal force and element load part to be changed to mass as long as it meets the requirements explained above.

The element loads will be lumped / distributed to the start and end node of the beam element according to the centre of gravity of the load.

CHANGE NODE

...	NODE	GROUP	...
		INTERSECTION	
		LINE	
		ROTATE	
		TRANSLATE	
		nodeno	

PURPOSE:

The command changes nodal coordinates.

Except for the ROTATE and TRANSLATE alternatives the command is very similar to the command for creating nodes, NODE; see this. The ROTATE and TRANSLATE alternatives are therefore described in more detail in the following.

NOTES:

Changing a node position after having defined the local coordinate system of an element may lead to error; see Section 3.3.3.

See also:

```
DELETE NODE ...
DISPLAY NODE ...
LABEL NODE-NUMBERS ...
LABEL NODE-SYMBOLS ...
LOAD NODE ...
NODE ...
PRINT NODE ...
RENUMBER NODE ...
```


CHANGE NODE ROTATE

...	ROTATE	trano	select-nodes
-----	--------	-------	--------------

PURPOSE:

The command rotates selected nodes thereby changing their coordinates. If all nodes are selected this is equivalent to making the transformed coordinate system trano the new global coordinate system. See also Section 3.10.1

PARAMETERS:

trano	Reference number of the transformed coordinate system defined by the TRANSFORMATION command.
select-nodes	Select elements; see Section 5.1.

NOTES:

Warning: Boundary conditions and loads are not rotated.

Note: Do not use CHANGE NODE ROTATE on nodes with connected elements which have been given specific local coordinate system (which also will have effect on hinges) or element eccentricities.

When rotating a complete model (CHANGE NODE ROTATE trano ALL) the alignment attributes must also be deleted prior to rotation. Use the command DELETE ALIGNMENT ALL.

See also:

TRANSFORMATION . . .

CHANGE NODE TRANSLATE

...	TRANSLATE	dx	dy	dz	select-nodes
-----	-----------	----	----	----	--------------

PURPOSE:

The command translates selected nodes thereby changing their coordinates. May e.g. be used to elevate all nodes in a deck without affecting other nodes. See also Section 3.10.1.

PARAMETERS:

dx

dz Translations in the x-, y- and z-directions

dy

select-nodes Select nodes; see Section 5.1.

CHANGE STUB

...	STUB	BRACE	node	element	dy	thk	sfy	sfz	length
		JOINT	node						

or if the SET ASSIGN-OPTION SECTION-NUMBER is switched ON:

...	STUB	BRACE	node	element	secno	length
		JOINT	node			

PURPOSE:

The command changes a stub section, either one of the incoming braces, or to all braces entering the joint. See Section 3.6.1 regarding the ASSIGN-OPTION switch.

PARAMETERS:

BRACE	Assign to selected brace.
JOINT	Assign to all braces in joint.
node	Node for start of stub section.
element	Element to modify.
dy	Pipe outer diameter (default = element to split).
thk	Thickness of pipe wall (default = element to split).
sfy	Pipe section shear area modifying factor, local y axis.
sfz	Pipe section shear area modifying factor, local z axis.
length	Stub length.
secno	Section number to be used as stub element.

NOTES:

For the JOINT option, the pipe section parameters must be given for each brace. The default stub length is calculated according to given parameters (see command SET CAN-STUB-LENGTH-PARAMETERS) and joint geometry.

If the pipe section parameters given do not correspond to an existing pipe section, a new section will automatically be created.

Elements belonging to a member may at any time be modified with respect to section and material by use of the command PROPERTY CONNECT.

See also:

ASSIGN STUB ...

SET CAN-STUB-LENGTH-PARAMETERS ...

PROPERTY CONNECT ...

COPY

COPY	LINE	source	destination	node-number-increment	element-number-increment
	PLANE				
	SET	setname	vector		

PURPOSE:

The command copies all previously created nodes and elements of the referred line segment, plane or set to a new position. The new node and element numbers are determined by incrementing the numbers of the source nodes/elements by user defined values.

The following data are copied:

- nodes,
- elements,
- cross sections of the elements, i.e. the reference to a cross section number,
- material properties of the elements, i.e. the reference to a material number,
- local coordinate systems of the elements in the case of copying a plane (not when copying a line),
- eccentricities (offsets) defined for the elements,
- hinges defined for the elements.

The following data are not copied:

- elements of type SPRING-TO-GROUND and DAMPER-TO-GROUND,
- nodal masses,
- boundary conditions,
- local coordinate systems of the elements in the case of copying a line,
- loads.

PARAMETERS:

source	Two (LINE option) or three (PLANE option) nodes identifying the nodes and elements to be copied.
destination	Two (LINE option) or three (PLANE option) nodes determining the position of the copy.
setname	A previously defined set name.

vector	A vector determining the position of the copy of a set of nodes and elements, from the source to the destination.
node-number-increment	An increment to apply to all nodes of the destination compared to the source. Care should be taken to avoid conflicting node numbers.
element-number-increment	An increment to apply to all elements of the destination compared to the source. Care should be taken to avoid conflicting element numbers.

NOTES:

For the LINE option the source nodes and elements are selected by referring to the two end nodes of the line segment. The destination line segment is defined by referring to nodes that already exist. Compared with the source, the destination may be contracted or extended. The distances between the nodes along the lines will be kept in proportion.

For the PLANE option the source nodes and elements are selected by referring to three nodes within the plane. The destination plane is defined by referring to nodes that already exist. Scaling of the source plane is not allowed; the two triangles defined by the two sets of nodes, must be congruent. Local coordinate systems and eccentricities are rotated in the same way as the nodes when copying a plane.

For the SET option the whole set referred to by name is copied to a destination identified by a vector from the source to the destination.

The copy command will not copy a node or an element to the same position as an existing node or element.

CREATE

CREATE	MEMBER	sub-commands
	MEMBER-FROM-ELEMENT	

PURPOSE:

The command creates members. A member is a modelling concept defined by a start and an end node (joints), and one or more elements (segments) between the two nodes.

Can and stub sections can only be the start or end segment of a member, hence members shall normally be defined between two structural joints. The member information is read and used by FRAMEWORK. (Reminder: FRAMEWORK is capable of handling joint related data (e.g. performing joint capacity check) on joints at member ends only.)

Members may be displayed by the command `DISPLAY MEMBER` and labelled by the command `LABEL MEMBER-NAMES`.

Elements belonging to a member are automatically assigned the alignment attribute.

PARAMETERS:

MEMBER	Manually create a member.
--------	---------------------------

MEMBER-FROM-ELEMENTS	Create members from selected elements.
----------------------	--

The sub-commands and data are fully explained on the following pages.

CREATE MEMBER

...	MEMBER	name	node1	node2
-----	--------	------	-------	-------

PURPOSE:

The command creates a member defined by element(s) on line a straight line between two joints / nodes.

PARAMETERS:

name	Member name.
node1	Existing node defining start of member.
node2	Existing node defining end of member.

NOTES:

When using double beam elements, e.g. jacket leg and inner pile connected by use of shim elements, please note the following. To avoid problems with respect to creating member concepts use offsets between the nodes for leg and nodes for pile. The offset (perpendicular to beam element local X axis) must be larger than the defined coordinate tolerance (ref. command SET COORDINATE-TOLERANCE value).

CREATE MEMBER-FROM-ELEMENT

...	MEMBER-FROM-ELEMENT	select-elements
-----	---------------------	-----------------

PURPOSE:

The command creates members by a one to one mapping between selected elements and members. The member name will be equal to the element number.

PARAMETERS:

select-elements Select elements by use of standard select element options.

DEFINE

...	GENSOD-DATA	sub-commands
	SET	
	SOIL	

PURPOSE:

The command defines data types as explained below.

PARAMETERS:

GENSOD-DATA	The command is used to define the additional data used on the GENSOD.INP file.
SET	The command defines a set of elements and/or nodes that may be referred to in commands where selecting elements or nodes is required.
SOIL	The command defines the soil profile and layer divisions.

The sub-commands and data are fully explained on the following pages.

DEFINE GENSOD-DATA

...	GENSOD-DATA	CONTROL	confrc	conlth	gammaw	atmprs	zcycl	sustif	
		LOAD-AT-SOIL-SURFACE	sigsrf	dpemb	aemb	bemb	dpcirc	radius	
		MATERIAL-COEFFICIENT	sftphi	sfsu	sfskf	sfsigt			
		PILE-DIAMETERS	esol0	esol1	posavr				
		SOIL-SURFACE	scrgen	scrloc	slope	zgrwt	gampwp		

PURPOSE:

The command is used to define the additional data used on the GENSOD.INP file. The data are connected to the following parts of the input file:

- CONTROL SECTION
- MATERIAL COEFFICIENTS SECTION
- PILE DIAMETERS AND GROUP EFFECTS SECTION
- SOIL SURFACE AND GROUND WATER SECTION
- LOADS AT SOIL SURFACE SECTION

PARAMETERS:

See bold text in notes below.

NOTES:

The parameters shown in bold below are defined by use of this command. (Default values are shown.) Note that the ZSURF parameter is set equal to mudline level but with opposite sign.

Input parameters referring to Z-LEVEL (i.e. the ZCYCL and ZGRWT parameters) shall be given Z values according to the superelement global coordinate system. (Negative value given as input will print out a positive value on the GENSOD.INP template file.)

If not given by the user, the ZCYCL and ZGRWT parameters are set equal to ZSURF.

```
***** CONTROL SECTION
1.000  CONFRC    OLD-FORCE-UNIT  = CONFRC * NEW-FORCE-UNIT  1MN  = 1000*1KN
1.000  CONLTH    OLD-LENGTH-UNIT = CONLTH * NEW-LENGTH-UNIT  1M   = 3.28*1FT
5      NUMTYP      NUMBER OF DIFFERENT SOIL TYPES
13     NUMTQZ      NUMBER OF LINES IN THE T-Z / Q-Z DATA TABLE BELOW
15     NUMLAY      NUMBER OF SOIL LAYERS
0      NUMDSP      NUMBER OF Z-LEVELS WITH GIVEN SOIL DISPLACEMENTS
```

```

1      MIDBOT      P-Y ETC COMPUTED AT : 1=LAYER-MIDPOINT 2=LAYER-BOTTOM
9.81  GAMMAW      UNIT WEIGHT OF WATER (9.81 KN/M3 IN SI-UNITS)
101.30 ATMPRS     ATMOSPHERIC PRESSURE (101.3 KN/M2 IN SI-UNITS)
ZSURF ZCYCL      Z-LEVEL DOWN TO WHICH CYCLIC P-Y DATA SHALL BE GENERATED
101.00 SUSTIF     USE STIFF CLAY P-Y PROCEDURES IF SU.GT.SUSTF (API ONLY)
2      JPRINT      PRINTED OUTPUT DATA (0=NONE 1=SOME 2=FULL)
1      JECHO       ECHO PRINT OF INPUT FILE NF14 TO FILE NF16 (0=NO 1=YES)

***** MATERIAL COEFFICIENTS SECTION
1.00   SFTPHI     MATERIAL COEFFICIENT ON TAN(PHI)
1.00   SFSU       MATERIAL COEFFICIENT ON UNDRAINED SHEAR STRENGTH
1.00   SFSKF      MATERIAL COEFFICIENT ON PILE SKIN FRICTION
1.00   SFSIGT     MATERIAL COEFFICIENT ON PILE TIP RESISTANCE

***** PILE DIAMETERS AND GROUP EFFECTS SECTION
2      NUMDIA      NUMBER OF PILE DIAM FOR WHICH P-Y/T-Z/Q-Z DATA IS WANTED
0.65  1.25        PILE DIAMETERS
10000.00 ESOL0    E-SOIL FOR GROUP EFFECT CALCULATION :
1200.00 ESOL1      ESOIL(Z) = ESOL0 + ESOL1*Z
0.50  POSAVR     SOIL AVERAGE POISSON RATIO FOR GROUP EFFECTS

***** SOIL SURFACE AND GROUND WATER SECTION
-(mudline) ZSURF   Z-LEVEL OF NON-SCoured SOIL SURFACE
2.00   SCRGEN     DEPTH OF GENERAL SCOUR BELOW ZSURF
4.00   SCRLOC     DEPTH OF LOCAL SCOUR BELOW ZSURF
20.00  SLOPE      SIDE SLOPE (DEGREES) OF LOCAL SCOUR HOLES
ZSURF  ZGRWT      Z-LEVEL OF GROUND WATER TABLE
9.81   GAMPWP     UNIT WEIGHT OF GROUND WATER (USED TO FIND PORE WATER PRSS)

***** LOADS AT SOIL SURFACE SECTION
0.00   SIGSRF     VERTICAL STRESS AT SURFACE
0.00   DPEMB      VERTICAL STRESS UNDER EMBANKMENT LOADING
0.00   AEMB       WIDTH A OF EMBANKMENT SLOPING PART
0.00   BEMB       PILE POSITION W.R.T. EMBANKMENT TOE (POSITIVE OUTSIDE)
0.00   DPCIRC     VERTICAL STRESS UNDER CIRCULAR LOADED AREA
0.00   RADIUS     RADIUS OF CIRCULAR LOADED AREA (PILE IS IN CENTER)
0      NUMFRC      NUMBER OF VERTICAL POINT FORCES AT SOIL SURFACE
0.00   POINT FORCE VALUES
0.00   HORIZONTAL DISTANCE TO PILE AXIS

```

This command may also be used to modify existing data.

The CONFRC and CONLTH parameters are also used on the SPLICE.INP template file.

See also the Gensod User Manual for specific soil related explanation.

DEFINE SET

...	SET	setname	INTERSECTION-WITH	...	ELEMENT	select-elements
			SUBTRACT-BY		NODE	select-nodes
			UNION-WITH			

PURPOSE:

The command defines a set of elements and/or nodes that may be referred to in commands where selecting elements or nodes is required. DEFINE SET defines a new set while CHANGE SET changes an existing set. The command syntaxes of these two commands are identical and based on standard set operators.

PARAMETERS:

setname	User-given name of the SET to define (maximum 8 characters and starting with a letter).
INTERSECTION-WITH	All elements and nodes except for those subsequently selected will be removed from the set. I.e. the new contents will be the intersection between the current contents of the set and the subsequent selection.
SUBTRACT-BY	The subsequently selected elements and nodes will be removed from the set.
UNION-WITH	The subsequently selected elements and nodes will be added to the set.
ELEMENTS	Elements are to be selected.
select-elements	Select elements; see Section 5.1.
NODES	Nodes are to be selected.
select-nodes	Select nodes; see Section 5.1.

NOTES:

Initially, i.e. after giving the command DEFINE SET and entering a name of the set, the set is empty. The first operation to do will therefore be to add to the set by selecting the UNION-WITH command. Thereafter, repetitive set operations may be performed until the content of the set is as desired. The operations are executed consecutively, the order of the operations is therefore of consequence. Finally, defining (or changing) the set is concluded by entering END rather than one of the set operators.

Sets defined are written to the Input Interface File and transferred through the linear analysis program Sestra to the postprocessors Framework and Postfem where they may be retrieved. Sets read from an Input Interface File (see the READ command) are available for further manipulations in Preframe.

EXAMPLES:

```
DEFINE SET SETA UNION NODE PLANE 11011 12011 13011 NO END
DEFINE SET SETB UNION ELEMENT PLANE 13011 14011 14061 NO END
DEFINE SET SETC UNION ELEMENT SET SETB NO
                        UNION NODE PLANE 13011 14011 14061 NO END
```

DEFINE SOIL

...	SOIL	PROFILE	soil-id	nofdiv	{ zbotm	noflay	soityp }*
		PARAMETER	MUDLINE-LEVEL	z-level			
			CURVE-FITTING-FACTOR	rftz			
			ZONE-OF-INFLUENCE	zoninf			

PURPOSE:

The command defines the soil profile and layer divisions. The command also defines the soil parameters used in connection with display of soil profile, generation of piles and writing of GENSOD input file.

PARAMETERS:

soil-id	Soil profile id (number). Currently only id 1 allowed.
nofdiv	Number of soil divisions (types) in profile.
zbotm	Z level (global co-ordinates) at bottom of division (type) n.
noflay	Number of layers within the soil division (type).
soityp	Soil type to be used in soil division.
MUDLINE-LEVEL	Define the Z level (global co-ordinates) for mudline.
z-level	The Z level defining mudline.
CURVE-FITTING-FACTOR	Define the curve fitting factor for TZ code = 200.
rftz	The value to be used for curve fitting (default = 0.9).
ZONE-OF-INFLUENCE	Define the (zone of influence)/(pile radius) for TZ code = 200.
zoninf	The value to be used for zone of influence (default = 10.0).

NOTES:

The {} indicates that the input command will loop and ask for input parameters for each soil division (nofdiv).

The soil profile Z levels must be given with increasing depth, i.e. increasing negative Z value.

This command may also be used to modify existing data.

See also the Gensod User Manual for specific soil related explanation, and Figure 4.1.3 in the Splice test example manual (SGP-EX).

DELETE

DELETE	ALIGNMENT	select-elements
	BOUNDARY-CONDITION	select-nodes
	ECCENTRICITY	select-elements
	ELEMENT	select-elements
	HINGE	select-elements
	INITIAL-CONDITION	...
	LINEAR-DEPENDENCY	select-nodes
	LOAD	...
	MASS-ON-NODE	select-nodes
	MATERIAL	matn
	MEMBER	ALL
		BY-ELEMENT
		BY-NAME
	NODE	select-nodes
	PILE-CONCEPT	...
	SECTION	sctn
	SET	setnam
	SOIL	...
	TRANSFORMATION	trano
	UNCONNECTED-NODES	

PURPOSE:

The commands deletes data previously defined. Only the DELETE INITIAL-CONDITION and the DELETE LOAD commands are treated in the following pages. Some notes are given below for some other alternatives. Otherwise see the defining commands.

PARAMETERS:

select-nodes	Select nodes; see Section 5.1.
select-elements	Select elements; see Section 5.1.
ALL	Delete all existing members.

BY-ELEMENT Delete existing member containing at least one of the selected elements. (Select elements by use of standard select element options.)

BY-NAME Delete members according to specified names (NO to end name list).

NOTES:

The DELETE ALIGNMENT deletes / removes any alignment attributes from selected elements.

The DELETE BOUNDARY-CONDITION command deletes boundary conditions, i.e. the selected nodes will then be FREE for all d.o.f.s. Note that the boundary conditions LINEAR and SUPERL (see the LINEAR-DEPENDENCY command) can only be deleted using the DELETE LINEAR-DEPENDENCY command.

The DELETE ELEMENT command deletes selected elements and related element loads. The number of d.o.f.s of the nodes are modified, if necessary, as explained for the ELEMENT command.

The DELETE LINEAR-DEPENDENCY command deletes linear dependencies between nodes. The linear dependencies of all selected dependent nodes are deleted. Note that the dependent nodes and not the independent nodes are to be given in this command. Also note that this command will delete the LINEAR boundary condition of the dependent node. It will also delete the SUPERL (or SUPER) boundary condition of the independent nodes unless other nodes still are linearly dependent of them.

The DELETE MATERIAL command deletes materials. The deletion of a material will remove all references to the material involving that some elements may no longer have a material property.

The DELETE MEMBER deletes / removes the conceptual member information connected to elements and nodes. This command does not delete the elements and nodes, only the conceptual information including conceptual attributes (i.e. hydrodynamic properties or stability parameters).

The DELETE NODE command deletes selected nodes. Elements connected to the deleted nodes will also be deleted. Loads defined for the explicitly deleted nodes and the implicitly deleted elements will also be deleted.

The DELETE PILE-CONCEPT command deletes nodes and elements used in pile concepts.

The DELETE SECTION command deletes cross sections. However, if a cross section is referred to by an element the program will refuse to delete the cross section.

The DELETE SET command deletes a named SET.

The DELETE SOIL command deletes SOIL DATA / DISPLAY / TYPE.

The DELETE TRANSFORMATION command deletes a transformation.

The DELETE UNCONNECTED-NODES command deletes all nodes which are not connected to any element.

DELETE INITIAL-CONDITION

...	INITIAL-CONDITION	DISPLACEMENT	select-nodes
		VELOCITY	
		BOTH	

PURPOSE:

The command deletes initial conditions of the nodes.

PARAMETERS:

DISPLACEMENT	Displacement option of initial condition is deleted.
VELOCITY	Velocity option of initial condition is deleted.
BOTH	Both options of initial conditions are deleted.
select-nodes	Select nodes; see Section 5.1.

NOTES:

See also:

```
CHANGE INITIAL-CONDITION ...
INITIAL-CONDITION ...
PRINT NODE INITIAL-CONDITION ...
```

DELETE LOAD

...	LOAD	load-case	...
-----	------	-----------	-----

...	ELEMENT	ALL	YES	select-elements		
			NO			
		DISTRIBUTED			select-elements	index
		POINT			select-elements	index
		CONSTANT-TEMPERAURE-ACROSS-THICKNESS			select-elements	index
	GRAVITY					
	NODE	ALL	YES	select-nodes		
			NO			
		FORCE			select-nodes	index
		PRESCRIBED-DISPLACEMENT			select-nodes	index
		PRESCRIBED-ACCELERATION			select-nodes	index
	ROTATION-OF-STRUCTURE					

PURPOSE:

The command deletes loads. See the LOAD command for a more detailed explanation of the load types.

The load index is used to distinguish between individual loads of the same type for the same node/element for the same load case. For example, a nodal force defined for the second time for the same node for the same load case is given index 2. Note that the load index may change after deletion; the load index always goes from 1 to N where N is the number of loads of the same type for that particular node/element.

PARAMETERS:

load-case	Load case number
ALL	Delete all loads.
YES/NO	Confirm deletion.
select-elements	Select elements; see Section 5.1.
DISTRIBUTED	Delete distributed element loads.
POINT	Delete element point loads.
index	Load index
CONSTANT-TEMPERATURE-ACROSS-THICKNESS	Delete element temperature loads.

GRAVITY	Delete gravity loads.
select-nodes	Select nodes; see Section 5.1.
FORCE	Delete nodal force loads.
PRESCRIBED-ACCELERATION	Delete nodal acceleration loads.
PRESCRIBED-DISPLACEMENT	Delete nodal displacement loads.
ROTATION-OF-STRUCTURE	Delete centripetal and tangential acceleration loads.

DELETE PILE-CONCEPT

...	PILE-CONCEPT	ALL	...	ALL
		CONDUCTOR		BY-ELEMENT
		MAIN		BY-NAME
		PILE-GROUP		

PURPOSE:

The command deletes nodes and elements used in pile concepts.

PARAMETERS:

ALL	All types / all occurrences.
CONDUCTOR	Only conductor piles.
MAIN	Only main piles.
PILE-GROUP	Only piles defined as pile groups.
BY-ELEMENT	Piles containing at least one of the selected elements. (Select elements by use of standard select element options.)
BY-NAME	Piles according to specified names (NO to end name list).

NOTES:

If deleting a node or element belonging to a pile, only the conceptual information connected to all elements and nodes belonging to that pile will be removed.

DELETE SOIL

...	SOIL	DATA	PY-TZ-QZ-CODE	...	NODE	node
			SKIN-FRICTION		Z-LEVEL	z-level
			TIP-RESISTANCE			
		DISPLAY				
		TYPE	soil-number			

PURPOSE:

The command deletes soil related data / display / type.

PARAMETERS:

DATA	Delete data-sets (PY-TZ-QZ codes, skin friction parameters and tip resistance parameters) for a given Z level.
PY-TZ-QZ-CODE	Delete PY-TZ-QZ codes.
SKIN-FRICTION	Delete skin friction parameters.
TIP-RESISTANCE	Delete tip resistance parameters.
NODE	Define the Z level by selecting a node.
node	The node defining the Z level.
Z-LEVEL	Define the z level by manually giving the z-value.
z-level	The z-value.
DISPLAY	Delete all nodes and elements used to display the soil profile.
TYPE	Delete a soil type.
soil-number	Number of soil type to be deleted.

NOTES:

Use the command PRINT SOIL to see defined DATA and TYPES.

The soil profile will automatically be delete prior to writing the SESAM Interface File (.FEM).

DISPLAY

DISPLAY	ELEMENT	select-elements		
	FOOTPRINT	nodeno		
	MEMBER	ALL		
		BY-ELEMENT		
		BY-NAME		
	JOINT	nodeno		
	NODE	select-nodes		
	PILE-CONCEPT	ALL	...	ALL
		CONDUCTOR		BY-ELEMENT
		MAIN		BY-NAME
		PILE-GROUP		

PURPOSE:

The command displays the model. See Section 3.16 for examples of displays.

The ELEMENT option causes the selected elements to be displayed alone, i.e. the nodes will not appear. The view may be zoomed and manipulated by the ROTATE command and SET GRAPHICS command. The LABEL command annotates the view.

The FOOTPRINT option displays the footprint of a joint, i.e. a developed view of a part of the chord with the brace intersections shown. The view cannot be zoomed or otherwise manipulated.

The MEMBER option displays the member concepts created.

The JOINT option displays a selected joint (node) with the elements coming into the node in more detail. The view may be zoomed and manipulated by the ROTATE command and SET GRAPHICS command. The LABEL command annotates the view.

The NODE option causes the selected nodes to be displayed (as yellow diamonds) plus the elements having all their nodes among the selected nodes. The view may be zoomed and manipulated by the ROTATE command and SET GRAPHICS command. The LABEL command annotates the view.

PARAMETERS:

select-elements	Select elements; see Section 5.1.
select-nodes	Select nodes; see Section 5.1.
nodeno	Selected node to be displayed.
ALL	Display all types / occurrences.

BY-ELEMENT	Display member / pile containing at least one of the selected elements. (Select elements by use of standard select element options.)
BY-NAME	Display members / pile according to specified names (NO to end name list).
CONDUCTOR	Display piles defined as conductor piles only.
MAIN	Display piles defined as main piles only.
PILE-GROUP	Display piles defined as group of piles only.

NOTES:

In silhouette and facettted display mode beam elements with GENERAL profile are displayed similar to BAR profiles but with dashed lines.

The profile height used is: $h = 2.0 * I_y / W_{ymin}$

The profile width used is: $b = 2.0 * I_z / W_{zmin}$

ELEMENT

ELEMENT	eltyp	GROUP
		LINE	node1	node2	
		elno	node1	node2	
		elno	nodeno		

PURPOSE:

The command creates elements described in Table 5.1.

Table 5.1 Element types and corresponding nodal properties

Element name	Number of nodes of the element	Number of d.o.f.s	Illustration in display and plot
BEAM-(BEAS)	2	6	
TRUSS-(TESS)	2	3	
NONSTRUCTURAL-BEAM-(BEAS/N)	2	6	
AXIAL-SPRING-(AXIS)	2	6	
AXIAL-DAMPER-(AXDA)	2	6	
SPRING-TO-GROUND-(GSPR)	1	1 to 6	
DAMPER-TO-GROUND-(GDAM)	1	1 to 6	
SHIM-ELEMENT-(GLSH)	2	1 to 6	
GENERAL-SPRING-(GLSH)	2	1 to 6	
PILE-SOIL-(PILS)	1	6	

The BEAM and TRUSS elements require material and cross sectional data which are defined (before or after creating the element) by the PROPERTY SECTION/MATERIAL commands and assigned to the elements by the PROPERTY CONNECT command subsequent to their creation. The NONSTRUCTURAL-BEAM only requires sectional data if wave loads are to be computed (by Wajac), and it will require both sectional data and material data (density only) if it shall contribute with mass to the structural analysis.

The AXIAL-SPRING, AXIAL-DAMPER, SPRING-TO-GROUND, DAMPER-TO-GROUND, SHIM-ELEMENT and GENERAL-SPRING elements only require material data. Note that in the case of the

SPRING-TO-GROUND and DAMPER-TO-GROUND elements, the material must be defined prior to creating the elements as the material number is referred to in the ELEMENT command (and consequently, no PROPERTY CONNECT command is required).

PARAMETERS:

eltyp	See Table 5.1 above.
GROUP	Create a group of elements as explained in the following pages.
LINE	Create a straight line of elements as explained in the following pages.
node1 node2	Nodes defining the straight line segment, alternatively the two nodes of the single two node element to create.
elno	Number of the single one node element to create.
nodeno	Node to which the one node element is connected.

NOTES:

Two node elements may be created individually, as a group of elements, or as a straight line of elements between two extreme nodes.

One node elements (SPRING- and DAMPER-TO-GROUND) may only be created individually.

The number of d.o.f.s of a node depends on the types of element connected to it: The number of d.o.f.s of a node is determined by the element with the highest number of d.o.f.s, and if this number is 1 or 2 it will be increased to 3, and if 4 or 5 it will be increased to 6. The reason for this increase is that the analysis programs (SESTRA) will only accept 3 or 6 d.o.f.s in a node.

Existing elements can be changed to be connected to other nodes by the CHANGE ELEMENT command. The element type cannot be changed.

The number of an element can be changed by the RENUMBER command.

Elements created may be deleted by the DELETE command.

The size of the one node element symbols are by default 20 mm (on the plot). The size may be modified by the SET GRAPHICS SIZE-SYMBOL ONE-NODED-ELEMENTS command.

The PILE-SOIL-(PILS) element is presently not in use.

Element numbers are limited to seven digits.

ELEMENT GROUP: BEAM, TRUSS, NONSTRUCTURAL-BEAM, AXIAL-SPRING, AXIAL-DAMPER, GENERAL-SPRING and SHIM-ELEMENT

...	GROUP	elno1	elno2	estep	node1	node2	nstep	coord-sys
-----	-------	-------	-------	-------	-------	-------	-------	-----------

where coord-sys is relevant only for the GENERAL-SPRING and SHIM-ELEMENT, and is:

GLOBAL			
LOCAL	elnor	X-Y-Z	newtrano
		Z-X-Y	
		Y-Z-X	
TRANSFORMATION	oldtrano		

PURPOSE:

The command creates a group of two node elements within a group of nodes. The node numbers must be organised in a such a way that there is a constant node numbering step between the first nodes of all elements to create and the same constant step between the second nodes. Figure 5.3 illustrates this.

PARAMETERS:

elno1	Number of the first element to create
elno2	Number of the last element to create
estep	Element numbering step = (elno2 - elno1) / (n-1), where n is the number of elements to create
node1 node2	The two nodes of element elno1
nstep	Node numbering step between the nodes of the elements to create. The two nodes (node1+nstep), (node2+nstep) are the nodes of element (elno1+estep).
GLOBAL	The coordinate system of the new element is the same as the global coordinate system.
LOCAL	The coordinate system of the new element is taken from the coordinate system of the previously created element elnor as follows: The X-Y-Z axes of elnor corresponds to the X-Y-X, Z-X-Y or Y-Z-X axes of the new element elno. The transformation from the global coordinate system to the local coordinate system of X-Y the new element is stored as a new transformation with reference number newtrano.
TRANSFORMATION	The coordinate system of the new element is the global coordinate system transformed with the previously defined transformation number oldtrano.

EXAMPLES:

ELEMENT BEAM GROUP 11 41 10 101 111 100

See Figure 5.3; the new elements 11 through 41 with step of 10, are created within the previously created group of nodes 101 through 411. The element type is BEAM and the nodes of the first element 11 is 101 and 111. The elements need not be in the same plane as in this case.

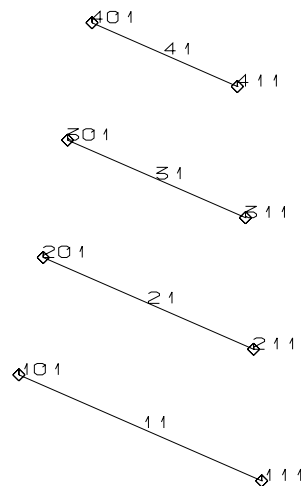


Figure 5.3 Creating an ELEMENT GROUP

ELEMENT LINE: BEAM, TRUSS, NONSTRUCTURAL-BEAM, AXIAL-SPRING, AXIAL-DAMPER, GENERAL-SPRING and SHIM-ELEMENT

...	LINE	node1	node2	elno*			coord-sys
				STEP	first-element	element-step	
				AUTO			

where coord-sys is relevant only for the GENERAL-SPRING and SHIM-ELEMENT, and is:

GLOBAL			
LOCAL	elnor	X-Y-Z	newtrano
		Z-X-Y	
		Y-Z-X	
TRANSFORMATION	oldtrano		

PURPOSE:

The command creates two node elements between nodes positioned on a straight line segment. There is no requirement to a constant step in node numbering along the line; all nodes on a straight line between the two nodes defining the line segment are taken into account. Figure 5.4 illustrates this.

PARAMETERS:

node1 node2	Nodes defining the straight line segment
elno*	Numbers of the elements to create. A number is manually assigned to each element.
STEP	Element numbers will be generated by a step function.
first-element	Number of the first element to create.
element-step	Step in the element numbering.
AUTO	The elements will be given numbers sequentially, starting with the currently highest element number used plus one.
GLOBAL	The coordinate system of the new element is the same as the global coordinate system.
LOCAL	The coordinate system of the new element is taken from the coordinate system of the previously created element elnor as follows: The X-Y-Z axes of elnor corresponds to the X-Y-X, Z-X-Y or Y-Z-X axes of the new element elno. The transformation from the global coordinate system to the local coordinate system of the new element is stored as a new transformation with reference number newtrano.

TRANSFORMATION The coordinate system of the new element is the global coordinate system transformed with the previously defined transformation number oldtrano.

EXAMPLES:

ELEMENT BEAM LINE 111 141 STEP 11 10

See Figure 5.3: the new elements 11 through 61 are created along the line segment defined by nodes 111 and 141. The nodes 111, 112, 113, 114, 121, 131 and 141 are positioned on a straight line. Note that the node numbering step is not constant. The program determines that there will be six elements along the line and demands six element numbers to be given.

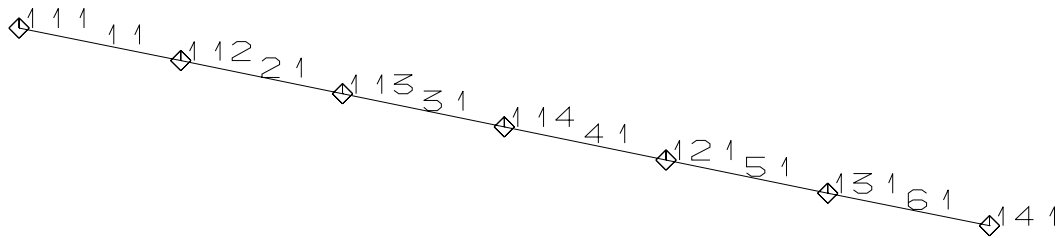


Figure 5.4 Creating an ELEMENT LINE

ELEMENT single: GENERAL-SPRING and SHIM-ELEMENT

...	elno	node1	node2	coord-sys
-----	------	-------	-------	-----------

where coord-sys is:

GLOBAL			
LOCAL	elnor	X-Y-Z	newtrano
		Z-X-Y	
		Y-Z-X	
TRANSFORMATION	oldtrano		

PURPOSE:

The command creates a GENERAL-SPRING or a SHIM-ELEMENT between two existing nodes.

PARAMETERS:

elno	Number of the element to create
node1 node2	The two nodes of element elno
GLOBAL	The coordinate system of the new element is the same as the global coordinate system.
LOCAL	The coordinate system of the new element is taken from the coordinate system of the previously created element elnor as follows: The X-Y-Z axes of elnor corresponds to the X-Y-X, Z-X-Y or Y-Z-X axes of the new element elno. The transformation from the global coordinate system to the local coordinate system of the new element is stored as a new transformation with reference number newtrano.
TRANSFORMATION	The coordinate system of the new element is the global coordinate system transformed with the previously defined transformation number oldtrano.

ELEMENT single: SPRING-TO-GROUND and DAMPER-TO-GROUND

...	elno	nodeno	GLOBAL			matno
			LOCAL	elnor	newtrano	
			TRANSFORMATION	oldtrano		

PURPOSE:

The command creates a SPRING-TO-GROUND or a DAMPER-TO-GROUND element connected to an existing single node.

PARAMETERS:

elno	Number of the element to create
nodeno	The node to which the new element is connected
GLOBAL	The coordinate system of the new element is the same as the global coordinate system.
LOCAL	The coordinate system of the new element is the same as the coordinate system of the previously created element elnor. The transformation from the global coordinate system to the local coordinate system of the new element is stored as a new transformation number newtrano.
TRANSFORMATION	The coordinate system of the new element is the global coordinate system transformed with the previously defined transformation number oldtrano.
matno	The material number referring to a SPRING-TO-GROUND stiffness or DAMPER-TO-GROUND damping matrix that previously must have been defined by the PROPERTY MATERIAL command.

EXAMPLES:

ELEMENT SPRING-TO-GROUND 15 111 GLOBAL 2

See Figure 5.5 (left); the new SPRING-TO-GROUND element is given number 15 and is connected to node 111 (node numbers are not labelled). The coordinate system of the element is the same as the global coordinate system. The material number, which previously must have been defined as a SPRING-TO-GROUND stiffness matrix, is 2.

ELEMENT DAMPER-TO-GROUND 16 111 TRANSFORMATION 5 3

See Figure 5.5 (right); the new DAMPER-TO-GROUND element is given number 16 and is connected to node 111 (node numbers are not labelled). The coordinate system of the element is the global coordinate system transformed with transformation number 5. The material number, which previously must have been defined as a DAMPER-TO-GROUND damping matrix, is 3.

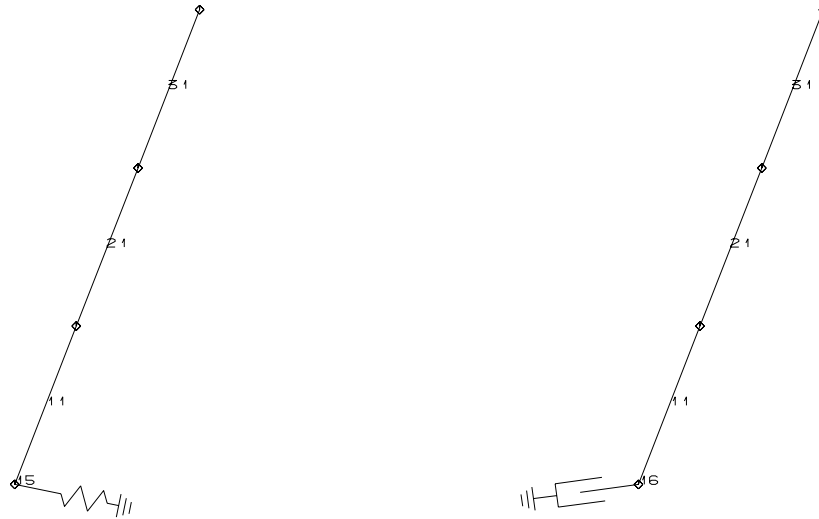


Figure 5.5 SPRING-TO-GROUND element (left) and a DAMPER-TO-GROUND element (right)

ELEMENT single: BEAM, TRUSS, NONSTRUCTURAL-BEAM, AXIAL-SPRING and AXIAL-DAMPER

...	elno	node1	node2
-----	------	-------	-------

PURPOSE:

The command creates a 2 node element connected to two existing nodes.

PARAMETERS:

elno	Number of element to create
node1 node2	The two nodes of element elno

EXIT

EXIT

PURPOSE:

The command interrupts the program execution. All files opened are properly saved and closed. The user may resume the modelling at a later stage by referring to the model file and command log file as 'old' when re-entering Preframe.

GENERATE

GENERATE	eltyp	JACKET	...
		K-BRACING	
		LINE	
		PILE	
		PILE-FROM-SOIL	
		T-BRACING	
		X-BRACING	

PURPOSE:

The GENERATE command creates several nodes and elements by a single command. The command substitutes repeated use of the NODE and ELEMENT commands. To create a jacket model for example, use the GENERATE BEAM JACKET command followed by other GENERATE commands to refine the model.

Options are available for creating nodes and elements for:

- JACKET — command creates all legs and the main bracings of a complete four-, six- or eight legged jacket model, this model may then be refined. The BEAM-(BEAS) type of element is normally the only relevant choice for eltyp (the command will optionally also create NONSTRUCTURAL-BEAM-(BEAS/N) elements; see below).
- K-BRACING — command creates a K-bracing by splitting an existing element into two new elements and inserting elements between the new node and two other nodes.
- LINE — command creates a line of nodes and elements between two existing nodes.
- PILE — command generates one or several piles (pile concepts) based on selection of node(s), or one pile based on a specific node and element.
- PILE-FROM-SOIL — command generates one or several piles (pile concepts) based on the soil profile / id given.
- T-BRACING — command creates a T-bracing by projecting an existing node onto an existing element, splitting the element into two new elements, and creating a new element connecting the existing node with the new node.
- X-BRACING — command creates an X-bracing between four existing nodes.

PARAMETERS:

eltyp Type of element, relevant element types are:
 BEAM-(BEAS), the regular two node beam,

NONSTRUCTURAL-BEAM-(BEAS/N), the non-structural beam, and
TRUSS-(TESS), the truss element.

NOTES:

Elements type BEAM-(BEAS) must be used in connection with the PILE options.

GENERATE eltyp JACKET

...	JACKET	4-LEGGED	l-bot	w-bot	z-bot	l-top	w-top	z-top	l-space	...
		6-LEGGED								
		8-LEGGED								

...	z-elev*	END	...
-----	---------	-----	-----

followed by defining bracing type, conductors, sections and possible top offset defined through:

...	BRACINGS	X-BRACINGS	row-no	...	elev-no*	END
			ALL-LONGITUDINAL-ROWS		ALL-ELEVATIONS	
			ALL-ROWS			
			TRANSVERSE-ROW			
			END			
		END				

...	CONDUCTORS	{x-cndct y-cndct}*							...	BEAMS
		REGULAR-GRID	xg	yg	nx	xsp	ny	ysp		NODES-ONLY
										NONSTRUC-
										TURAL
END										

...	SECTIONS	CONDUCTORS			...	sctno
		HORIZONTAL-BRACINGS	elev-no			NONE
			ALL-ELEVATIONS			
			END			
		LEGS	elev-no			
			ALL-ELEVATIONS			
			BOTTOM-ELEVATION			
			END			
		X-BRACINGS	elev- no			
			ALL-ELEVATIONS			
			END			
		END				

...	TOP-OFFSET	x-offset	y-offset
-----	------------	----------	----------

PURPOSE:

The command creates all legs and the main bracings of a four-, six- or eight legged jacket model. The model may contain an arbitrary number of horizontal bracings (elevations) and it may also be given a top offset yielding a askew jacket. This model may then be refined as required by creating additional bracings, changing section assignments, etc.

The origin of the coordinate system of the jacket will, when seen from above, be in the centre of the jacket. The X-axis will be in the length direction, the Y-axis in the width direction, and the Z-axis will point upwards. The vertical position of the origin will indirectly be determined by the Z-coordinate given for the bottom.

The principle of the command is:

- give the main dimensions like length, width and Z-coordinate of the bottom and top of the jacket,
- specify the elevations (Z-coordinates) of the horizontal bracings,
- select additional bracing (presently limited to X-bracing) by referring to row and elevation numbers,
- optionally, define vertical conductors by giving their X- and Y-coordinates,
- assign section numbers to the various elevations of the legs, bracings and conductors, and
- specify possible top offset of the jacket.

An exemplified guide in how to use this command is found in Section 3.2.4. The user is advised to refer to this section to learn about practical use of the command.

PARAMETERS:

4-LEGGED	Generate a four legged jacket.
6-LEGGED	Generate a six legged jacket.
8-LEGGED	Generate an eight legged jacket where the two middle legs are parallel launch legs.

Main dimensions:

l-bot w-bot z-bot	The length, width and Z-coordinate (in the model's cartesian coordinate system) of the jacket at the bottom of the legs.
l-top w-top z-top	The length, width and Z-coordinate (in the model's cartesian coordinate system) of the jacket at the top of the legs.
l-space	The spacing between the launch legs of an 8-LEGGED jacket. The spacing is constant, i.e. the launch legs are parallel even though the corner legs are not. This parameter will not be requested for 4- and 6-LEGGED jackets.

Elevations (Z-coordinates) of horizontal bracings:

z-elev*

Z-coordinates (in the model's cartesian coordinate system) of the elevations, i.e. the vertical positions of the horizontal bracings. The number of elevations is determined by the number of z-elev's entered before concluding with END. The first z-elev given must be equal to or greater than z-bot, if equal, then the bottom horizontal bracing will be at the bottom of the legs. The last z-elev given must be equal to or less than z-top, if equal, then the top horizontal bracing will be at the top of the legs.

Additional bracing:

BRACINGS

Select type of bracing.

X-BRACINGS

X-bracings will be inserted for rows and elevations as specified by the subsequent data.

row-n

Row number in longitudinal direction for which bracing will be inserted for given elevations.

ALL-LONGITUDINAL-ROWS

Bracing will be inserted for all longitudinal rows for given elevations.

ALL-ROWS

Bracing will be inserted for all rows, in transverse as well as longitudinal directions, for given elevations.

TRANSVERSE-ROW

Bracing will be inserted for the transverse row for given elevations.

elev-no*

Elevation numbers for which bracing will be inserted.

ALL-ELEVATIONS

Bracing will be inserted for all elevations.

Conductors:

CONDUCTORS

Define vertical conductors according to the subsequent data.

{x-cndct y-cndct}*

X- and Y-coordinates (in the model's cartesian coordinate system) of individually given conductors.

REGULAR-GRID xg yg nx xsp ny ysp

Specify a conductor grid by giving the X- and Y-coordinates of a corner (xg and yg), the number of conductors and spacing between them in the X- and Y-directions respectively (nx, xsp and ny, ysp).

BEAMS

Let the conductors be represented by beam elements.

NODES-ONLY

Let the conductors be represented by nodes only. This alternative is relevant if the conductors are defined as a separate superelement.

NONSTRUCTURAL

Let the conductors be represented by non-structural beams.

Assign section numbers:

SECTIONS	Assign section numbers to elements created.
CONDUCTORS	
HORIZONTAL-BRACINGS	
X-BRACINGS	
LEGS	Assign section numbers to the conductors/horizontal bracings/legs/X-bracings accordingly.
elev-n	Elevation number at which the subsequently given section number applies.
ALL-ELEVATIONS	The subsequently given section number applies to all elevations.
BOTTOM-ELEVATION	The subsequently given section number applies to the bottom elevations, i.e. the elevation between the bottom of the jacket and the lowest horizontal bracing.
sctno	Section number that previously must have been defined.
NONE	No section number is assigned.
Possible top offset:	
TOP-OFFSET	The jacket has a horizontal offset of the centre of the jacket at top (at z-top) compared to the centre of the jacket at bottom (at z-bot). See the notes below on the effect of an offset on possible conductors.
x-offset y-offset	The horizontal offset of the top of the jacket.

NOTES:

For efficient use of this command define first the appropriate sections and refer to these within the command. Otherwise, the appropriate sections must be assigned (connected) afterwards.

Note that $X=0$ and $Y=0$ (in the model's cartesian coordinate system) will be in the centre of the jacket. The vertical position of the origin is determined by the Z-coordinate given for the bottom of the legs. The Z-coordinate of the top of the legs will implicitly define the height of the jacket.

Even if an offset of the top of the jacket is specified, i.e. the jacket is askew, the conductors (if defined) will still be vertical (each conductor is positioned by only one set of X- and Y-coordinates).

When using the CONDUCTORS alternative to create conductors a set named CONDUCT will automatically be created containing the nodes and elements belonging to the conductors. This set may be used in connection with making a separate superelement for the conductors; see Section 3.2.4 for more information on this.

Node and element numbering

The GENERATE BEAM JACKET command employs a system for numbering the nodes and elements. The numbering system for the nodes is as follows (also see the example of Figure 5.6):

- Nodes along the legs have five digit numbers beginning with 1. They can be written as $1ij1$, where:
 - i is a two digit number identifying the elevation number, 00 at the bottom of the legs, 01 at the first elevation, 02 at the second elevation, and so on, and
 - j is the leg number: 1-4 for four legged, 1-6 for six legged, and 1-8 for eight legged jackets.
- Nodes of the X-bracings have six digit numbers beginning with 3. They can be written as $3ijk1$, where:
 - i is as above: a two digit number identifying the elevation number, and
 - j and k are the leg numbers on either side of the node.
- Conductor nodes have six digit numbers beginning with 5. They can be written as $5im1$, where:
 - i is as above: a two digit number identifying the elevation number, and
 - m is a two digit number identifying the conductor: 01, 02, 03, etc.
- Note that all node numbers have 1 as last digit to allow for new node numbers to be inserted in-between.

The numbering system for the elements is as follows:

- Elements along the legs have five digit numbers beginning with 1. They can be written as $1ij1$, where:
 - i is a two digit number identifying the elevation number, 00 at the bottom of the legs, 01 at the first elevation, 02 at the second elevation, and so on, and
 - j is the leg number: 1-4 for four legged, 1-6 for six legged, and 1-8 for eight legged jackets.
- Elements of the horizontal bracings have six digit numbers beginning with 2. They can be written as $2ijk1$, where:
 - i is as above: a two digit number identifying the elevation number, and
 - j and k are the leg numbers on either side of the element.

Notice that the sequence of the leg numbers j and k also identifies in which direction the local x-axis is pointing, i.e. from j to k .
- Elements of the X-bracings have six digit numbers beginning with 3. The four elements constituting the X-bracing have numbers that can be written as $3ijk1$, $3ijk2$, $3ikj1$ and $3ikj2$, where:
 - i is as above: a two digit number identifying the elevation number, and
 - j and k are the leg numbers on either side of the element.

Notice that the sequence of the leg numbers j and k also identifies in which direction the local x-axis is pointing, i.e. from j to k .
- Conductor elements have six digit numbers beginning with 5. They can be written as $5im1$, where:
 - i is as above: a two digit number identifying the elevation number, and
 - m is a two digit number identifying the conductor: 01, 02, 03, etc.
- Notice that all element numbers have 1 or 2 as last digits to allow for new element numbers to be inserted in-between.

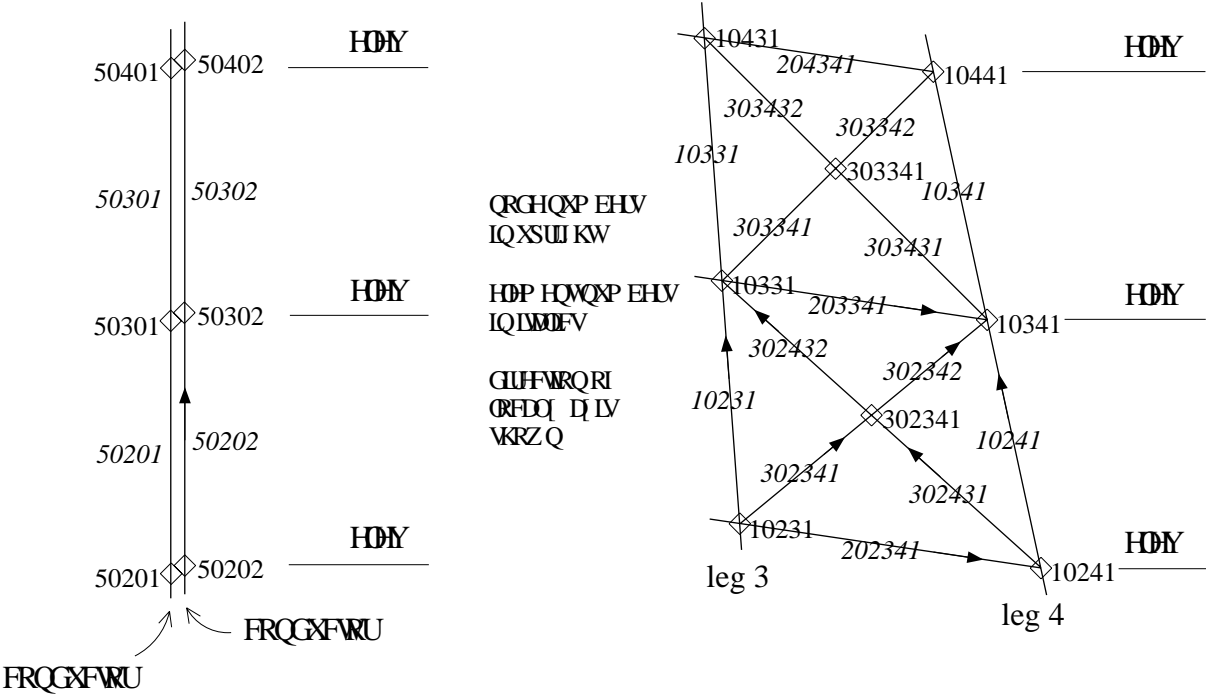


Figure 5.6 Illustration of the node and element numbering system, and local x-axis direction

GENERATE eltyp K-BRACING

...	K-BRACING	node1	node2	element	rel-pos	...
...	nodeno	...	elno*3			
	AUTO		STEP	first-element	element-step	
			AUTO			

PURPOSE:

The command creates an K-bracing by splitting an existing element into two new elements and connecting the new node with two other nodes. See Figure 5.7.

The element number of the element being split is given to the one of the two new replacing elements that is connected to end 1 of the original element (see NOTES below on how to determine which end is end 1). Therefore, three new element numbers are required and one new node number.

PARAMETERS:

node1 node2	First and second node.
element	Element number to split.
rel-pos	Relative position of the new node along the element to split, measured from end 1 of the element, i.e. in the direction of the local x-axis.
nodeno	Node number of the splitting node created.
AUTO	Node/element numbers will be generated automatically. automatically generated node number will be the highest current node number plus 1.three automatically generated element numbers will be element incremented by 1, 2 and 3. If these numbers are occupied by other elements then the element numbers will be the highest current element number plus 1, 2 and 3.
elno*3	Element numbers of the three elements created.
STEP	Element numbers will be generated step-wise.
first-element	Element number of first created element.
element-step	The step in element numbering.

NOTES:

Possible loads defined for the element being split are deleted.

You may find it convenient to display a panel of the model (DISPLAY ELEMENT PLANE) and then insert K-bracings by clicking the appropriate nodes and elements. The SET DEFAULT SECTION command may be used to pre-select the appropriate section for the K-bracings. Note that only the K-bracing will be

assigned this default section, the two elements replacing the split element will inherit the section of the original element.

Use LABEL LOCAL-COORDINATE to see which end is end 1: the local y- or z-axis (according to your choice) is drawn close to end 1.

Note that the SET NUMBERING-AUTOMATIC command may be used to switch on automatic assignment of node and element numbers. The command will then not request node and element numbers.

An eccentricity of the element to split is maintained by positioning the new node on the eccentric element.

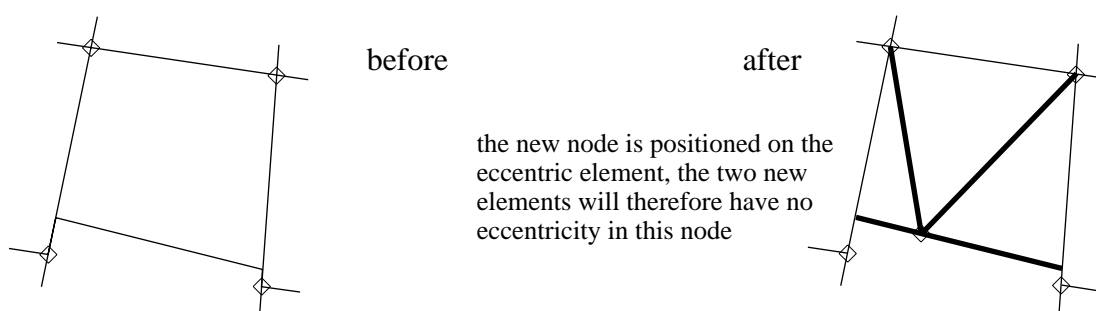


Figure 5.7 K-bracing when eccentricity

EXAMPLES:

Example of command for generating a K-bracing:

```
GENERATE BEAM-(BEAS) K-BRACING
201 211 10 0.3
105 11 15 16
END END
```

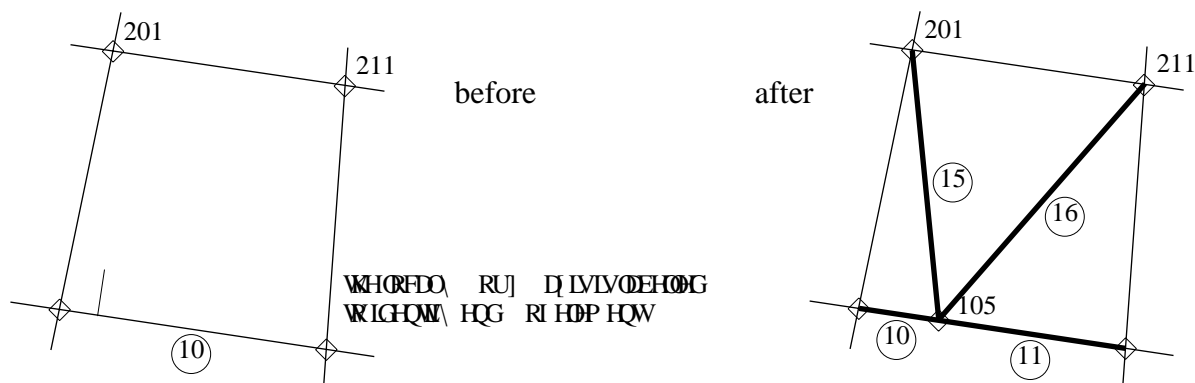


Figure 5.8 K-bracing created by GENERATE command

GENERATE eltyp LINE

...	LINE	node1	node2	ndiv	...				
...	nodeno*			...	elno*			...	EVEN
	STEP	first-node	node-step		STEP	first-element	element-step		space*
	AUTO				AUTO				

PURPOSE:

The command creates nodes and elements distributed along a straight line between two existing nodes. The direction (defines the local x-axis) of the line is from node1 to node2.

PARAMETERS:

node1 node2	The two existing nodes defining the line segment.
ndiv	Number of divisions of the line segment. ndiv-1 number of nodes and ndiv elements will be created.
nodeno*	Node numbers of the created nodes. Each node is manually given a number.
elno*	Element numbers of the created elements. Each element is manually given a number.
STEP	Node/element numbers will be generated step-wise.
first-node	Node number of first created node.
node-step	The step in node numbering.
first-element	Element number of first created element.
element-step	The step in element numbering.
AUTO	Node/element numbers will be generated automatically. The numbers will be generated in sequence starting with the highest current node/element number plus one.
EVEN	The line will be divided into ndiv equal spacings.
space*	The spacings between the nodes starting in node1. If all ndiv spacings are entered they will be interpreted as relative spacings. If less than ndiv spacings are entered, they will be interpreted as true spacings and the remaining part will be divided into equal spacings.

NOTES:

The SET DEFAULT SECTION command may be used to pre-select the appropriate section of the elements created.

Note that the SET NUMBERING-AUTOMATIC command may be used to switch on automatic assignment of node and element numbers. The command will then not request node and element numbers.

GENERATE eltyp PILE

...	PILE	CONDUCTOR	...	BY-NODE-SELECT	node-select		...
		MAIN		ONE-BY-ONE	node	element	
		PILE-GROUP					

...	nofseg	{ seglen	nofelem	sectno	matno }	name
-----	--------	----------	---------	--------	---------	------

PURPOSE:

The command generates one or several piles (pile concepts) based on selection of node(s), or one pile based on a specific node and element. When using the BY-NODE-SELECT option, it is presumed that it is only one element connected to each of the selected nodes. The piles will be generated in the opposite direction of the incoming / reference element.

PARAMETERS:

CONDUCTOR	Piles will be defined as conductor piles in the concept definitions.
MAIN	Piles will be defined as main piles in the concept definitions.
PILE-GROUP	Piles will be defined as group of piles in the concept definitions.
BY-NODE-SELECT	Create one or several equal piles based on selected node(s).
ONE-BY-ONE	Create one pile based on selected node and element. This option must be used if more than one element is connected to the reference node.
node-select	Select reference nodes for pile heads by use of standard select node options.
node	Reference node for pile head.
element	Element defining opposite direction of pile direction.
nofseg	Number of pile segments.
seglen	Total length of pile segment (loops for nofseg).
nofelem	Number of elements in pile segment (loops for nofseg).
sectno	Section number for elements in pile segment (loops for nofseg).
matno	Material number for elements in pile segment (loops for nofseg).
name	The pile name (to be given for the ONE-BY-ONE option only).

NOTES:

When using the BY-NODE-SELECT option, the pile name will be Pxxxxx, where xxxxx is the node number of the reference node. (When using the ONE-BY-ONE option, the default proposed pile name is Pxxxxx.)

Nodes and elements forming a pile will be assigned the following node and element numbers: $900000 + (100 * n) + \text{xxxxx}$, where xxxxx is the node number of the reference node, and n runs from 0 to N-1, where N is total number of nodes / elements generated to represent the pile.

In addition to the reference node given when creating the pile concepts, a pile head node will be defined. The pile head node will get an offset equal to 1/100 of the pile outer diameter from the reference node. Do not connect a section (PROPERTY CONNECT SECTION) later in the design process with smaller outer diameter than the original diameter. Test routines in Splice will then fail.

The piles are not allowed to interfere with other parts off the structure.

When creating pile groups, remember to assign the 'fixed-to' node attribute (i.e. the node to which all piles in the pile group shall be rigidly connected, see command ASSIGN PILE-DATA FIXED-TO). The 'fixed-to' node reference used for pile group must have equal Z-coordinate as the pile heads.

A pile segment is a part of a pile where all elements forming the segment have equal section number and material number. Each element within a pile segment have equal length ($\text{length} = \text{seglen} / \text{nofelem}$).

All elements belonging to a pile are automatically assigned alignment attributes, i.e. if the node in the pile tip is moved, all intermediate node will be moved to keep the elements forming the pile on a straight line.

A tip: Use separate material number(s) for the piles. The yield strength (ref. command ASSIGN PILE-DATA YIELD-STRENGTH) assigned to the pile concepts will in Framework be assigned to the material number connected to the pile elements.

GENERATE eltyp PILE-FROM-SOIL

...	PILE-FROM-SOIL	soil-id	CONDUCTOR	...	BY-NODE-SELECT	node-select		...
			MAIN		ONE-BY-ONE	node	element	
			PILE-GROUP					

...	z-tip	sectno	matno	name
-----	-------	--------	-------	------

PURPOSE:

The command generates one or several piles (pile concepts) based on the soil profile / id given. When using the BY-NODE-SELECT option, it is presumed that it is only one element connected to each of the selected nodes. The piles will be generated in the opposite direction of the incoming / reference element.

PARAMETERS:

soil-id	Soil profile identity to be used.
CONDUCTOR	Piles will be defined as conductor piles in the concept definitions.
MAIN	Piles will be defined as main piles in the concept definitions.
PILE-GROUP	Piles will be defined as group of piles in the concept definitions.
BY-NODE-SELECT	Create one or several equal piles based on selected node(s).
ONE-BY-ONE	Create one pile based on selected node and element. This option must be used if more than one element is connected to the reference node.
node-select	Select reference nodes for pile heads by use of standard select node options.
node	Reference node for pile head.
element	Element defining opposite direction of pile direction.
z-tip	Z level (global co-ordinates) defining the pile tip.
sectno	Section number for elements in pile.
matno	Material number for elements in pile.
name	The pile name (to be given for the ONE-BY-ONE option only).

NOTES:

One pile node will be generated in the middle of each soil layer defined in the soil profile (Ref. command DEFINE SOIL PROFILE).

When using the BY-NODE-SELECT option, the pile name will be Pxxxxx, where xxxxx is the node number of the reference node. (When using the ONE-BY-ONE option, the default proposed pile name is Pxxxxx.)

Nodes and elements forming a pile will be assigned the following node and element numbers: $900000 + (100 * n) + \text{xxxxx}$, where xxxxx is the node number of the reference node, and n runs from 0 to N-1, where N is total number of nodes / elements generated to represent the pile.

In addition to the reference node given when creating the pile concepts, a pile head node will be defined. The pile head node will get an offset equal to 1/100 of the pile outer diameter from the reference node. Do not connect a section (PROPERTY CONNECT SECTION) later in the design process with smaller outer diameter than the original diameter. Test routines in Splice will then fail.

The piles are not allowed to interfere with other parts off the structure.

When creating pile groups, remember to assign the 'fixed-to' node attribute (i.e. the node to which all piles in the pile group shall be rigidly connected, see command ASSIGN PILE-DATA FIXED-TO). The 'fixed-to' node reference used for pile group must have equal Z-coordinate as the pile heads.

All elements belonging to a pile are automatically assigned alignment attributes, i.e. if the node in the pile tip is moved, all intermediate nodes will be moved to keep the elements forming the pile on a straight line.

A tip: Use separate material number(s) for the piles. The yield strength (ref. command ASSIGN PILE-DATA YIELD-STRENGTH) assigned to the pile concepts will in Framework be assigned to the material number connected to the pile elements.

GENERATE eltyp T-BRACING

...	T-BRACING	node	element	...	
...	nodeno	...	elno*2		
	AUTO		STEP	first-element	element-step
			AUTO		

PURPOSE:

The command creates a T-bracing by projecting an existing node onto an existing element, splitting the element into two new elements, and creating a new element connecting the existing node with the new node. See Figure 5.9.

The element number of the element being split is given to the one of the two new replacing elements that is connected to end 1 of the original element (see NOTES below on how to determine which end is end 1). Therefore, two new element numbers are required and one new node number.

PARAMETERS:

node	Node to project onto an element.
element	Element onto which the node is projected.
nodeno	Node number of the created node.
AUTO	Node/element numbers will be generated automatically. automatically generated node number will be the highest current node number plus 1.two automatically generated element numbers will be element incremented by 1 and 2. If these numbers are occupied by other elements then the element numbers will be the highest current element number plus 1 and 2.
elno*2	Element numbers of the created elements. Both elements are manually given a number.
STEP	Element numbers will be generated step-wise.
first-element	Element number of first created element.
element-step	The step in element numbering.

NOTES:

Possible loads defined for the element being split are deleted.

You may find it convenient to display a panel of the model (DISPLAY ELEMENT PLANE) and then position the T-bracing by clicking the appropriate nodes and element. The SET DEFAULT SECTION command may be used to pre-select the appropriate section for the T-bracings. Note that only the T-bracing will be

assigned this default section, the two elements replacing the split element will inherit the section of the original element.

Use LABEL LOCAL-COORDINATE to see which end is end 1: the local y- or z-axis (according to your choice) is drawn close to end 1.

Note that the SET NUMBERING-AUTOMATIC command may be used to switch on automatic assignment of node and element numbers. The command will then not request node and element numbers.

An eccentricity of the element to split has the effect that T-bracing is not perpendicular to the split element.

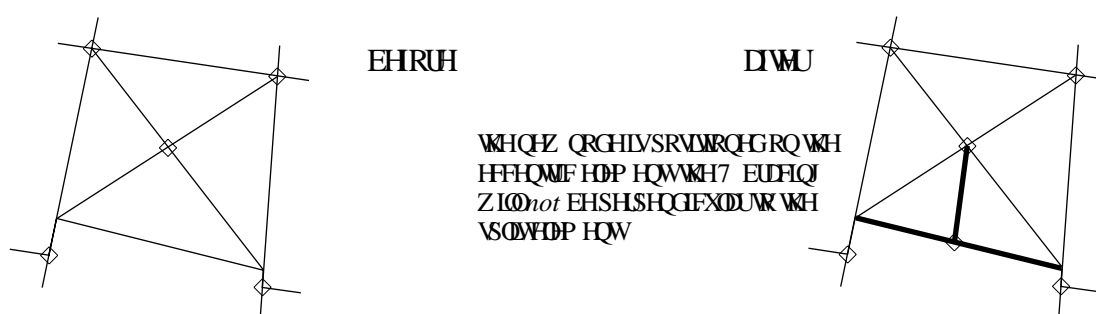


Figure 5.9 T-bracing when eccentricity

EXAMPLES:

Example of command for generating a T-bracing:

```
GENERATE BEAM-(BEAS) T-BRACING
151 10
105 11 15
END END
```

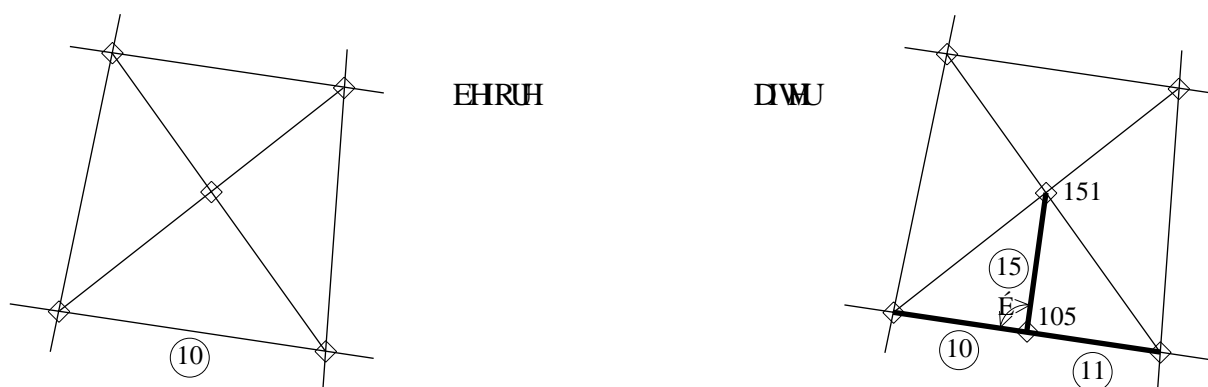


Figure 5.10 T-bracing created by GENERATE command

GENERATE eltyp X-BRACING

...	X-BRACING	node1	node2	node3	node4	nodeno	...
...	elno*4						
	STEP	first-element	element-step				
	AUTO						

PURPOSE:

The command creates an X-bracing between four existing nodes by creating a new node, nodeno, at the point of intersection between two straight lines defined by node1 - node2 and node3 - node4, and creating four new elements in the following sequence:

element 1: from node1 to nodeno

element 2: from nodeno to node2

element 3: from node3 to nodeno

element 4: from nodeno to node4

See Figure 5.11.

PARAMETERS:

node1 node2	First and second node of the first line.
node3 node4	First and second node of the second line.
nodeno	Node number of the intersection node created.
elno*4	Element numbers of the four elements created.
STEP	Element numbers will be generated step-wise.
first-element	Element number of first created element.
element-step	The step in element numbering.
AUTO	Element numbers will be generated automatically. The numbers will be generated in sequence starting with the highest current element number plus one.

NOTES:

You may find it convenient to display only a panel of the model by the DISPLAY ELEMENT PLANE command and then position the X-bracing by clicking the appropriate nodes. Further, the SET DEFAULT SECTION command may be used to pre-select the appropriate section of a number of X-bracings to be inserted.

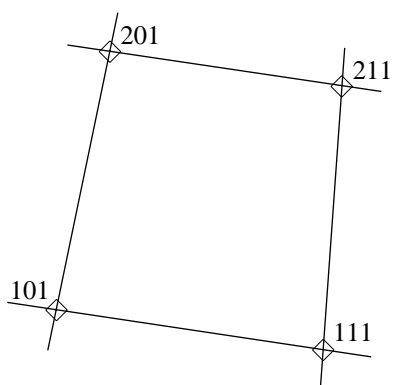
Note that the SET NUMBERING-AUTOMATIC command may be used to switch on automatic assignment of node and element numbers. The command will then not request node and element numbers.

The existence of element eccentricities has no effect on the X-bracing as the X-bracing is based on the position of the four nodes only.

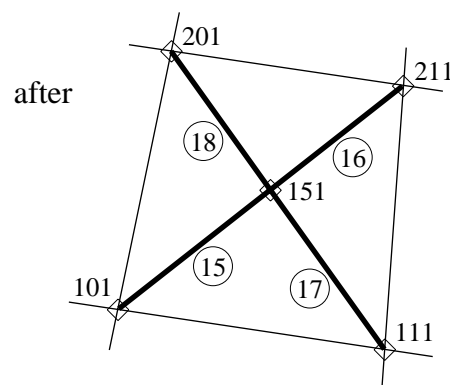
EXAMPLES:

Examples of command for generating a X-bracing

```
GENERATE BEAM- (BEAS) X-BRACING
  101 211 111 201
  151 15 16 17 18
END END
```



before



after

Figure 5.11 X-bracing created by GENERATE command

HELP

HELP	SUPPORT
	GENERAL-SYNTAX
	SPECIAL-KEYS
	STATUS-LIST

PURPOSE:

The command provides information on subjects. The information is printed in the line-mode window (message window for Windows NT).

PARAMETERS:

SUPPORT	The telephone and telefax numbers and the Internet address for requesting support is printed together with detailed information on the program version used. This information is of interest in connection with support requests.
GENERAL-SYNTAX	Information on how to enter commands and text is provided.
SPECIAL-KEYS	Information on some special keys is provided.
STATUS-LIST	If the program is used in line input mode the status list is printed on the screen. the program is used in graphical user interface the STATUS program is opened as a separate window with a graphical user interface.

INITIAL-CONDITION

INITIAL-CONDITION	incono	select-nodes	DISPLACEMENT	disp1	disp2...disp6
			VELOCITY	velo1	velo2...velo6

PURPOSE:

The command defines initial conditions of selected nodes in terms of displacements and/or velocities at time $t=0$ in a forced response analysis by time integration. An initial condition including both displacements and velocities may be assigned to a node simply by re-entering the command and referring to the same initial condition number. For data not defined the value 0.0 will be assumed.

Initial conditions may be changed by the CHANGE INITIAL-CONDITION command, and deleted by the DELETE INITIAL-CONDITION command.

PARAMETERS:

incono	Initial condition number
select-nodes	Select nodes; see Section 5.1.
DISPLACEMENT	Initial condition in terms of nodal displacements
VELOCITY	Initial condition in terms of nodal velocities
disp1...disp6	Displacements for the three translational and three rotational d.o.f.s.
velo1...velo6	Velocities for the three translational and three rotational d.o.f.s.

NOTES:

The initial conditions may only be defined when the model created by Preframe is the complete model, i.e. it contains no super d.o.f.s.

The initial condition feature is limited to only one set of displacements and/or velocities for each node.

LABEL

LABEL	BOUNDARY-CONDITION-SYMBOL	
	CONCEPT-ATTRIBUTES	sub-commands
	ELEMENT-NUMBERS	
	LOCAL-COORDINATE	LOCAL-Y-AXIS
		LOCAL-Z-AXIS
	MATERIAL-NUMBERS	
	MEMBER-NAMES	
	NODE-NUMBERS	EXTERNAL-NODE-NUMBER
		INTERNAL-NODE-NUMBER
	NODE-SYMBOLS	ALL-NODES
		SUPER-NODES-ONLY
	ORIGIN-SYMBOL	
	PILE-NAMES	
	SECTION-NUMBERS	
	SOIL-DATA	sub-commands

PURPOSE:

The command adds (labels) node symbols, node numbers, names etc. to the display. The labels are shown until a new display is made, the LABEL command may then be re-entered. The size of the symbols may be adjusted by the SET GRAPHICS SIZE-SYMBOLS command. The symbols used are shown in Figure 5.12.

PARAMETERS:

BOUNDARY-CONDITION-SYMBOL	Add symbols showing fixed d.o.f.s; see Figure 5.12.
CONCEPT-ATTRIBUTES	Add concept attributes, see following sub-command.
ELEMENT-NUMBERS	Add element numbers, do not use together with labelling of material numbers and section numbers as they will superimpose each other.
LOCAL-COORDINATE	Add either the elements' local y- or z-axes, both may be added by entering both commands.
LOCAL-Y-AXIS	Add either the elements' local y- or z-axes, both may be added by entering both commands.

LOCAL-Z-AXIS	Add either the elements' local y- or z-axes, both may be added by entering both commands.
MATERIAL-NUMBERS	Add material numbers, do not use together with labelling of element numbers and section numbers as they will superimpose each other.
MEMBER-NAMES	Add member names.
NODE-NUMBERS	Add the user defined external or program assigned internal node numbers. Normally, the user is only interested in the external node number.
EXTERNAL-NODE-NUMBER	Add the user defined external or program assigned internal node numbers. Normally, the user is only interested in the external node number.
INTERNAL-NODE-NUMBER	Add the user defined external or program assigned internal node numbers. Normally, the user is only interested in the external node number.
NODE-SYMBOLS	Add symbols for nodes; see Figure 5.12.
ALL-NODES	Add symbols for nodes; see Figure 5.12.
SUPER-NODE-ONLY	SUPER-NODE-ONLY
ORIGIN-SYMBOL	Add origin symbol.
PILE-NAME	Add pile names.
SECTION-NUMBER	Add section numbers, do not use together with labelling of element numbers and material numbers as they will superimpose each other.
SOIL-DATA	Add labels on the display of the soil profile.

NOTES:

See also:

SET GRAPHICS SIZE-SYMBOLS ...

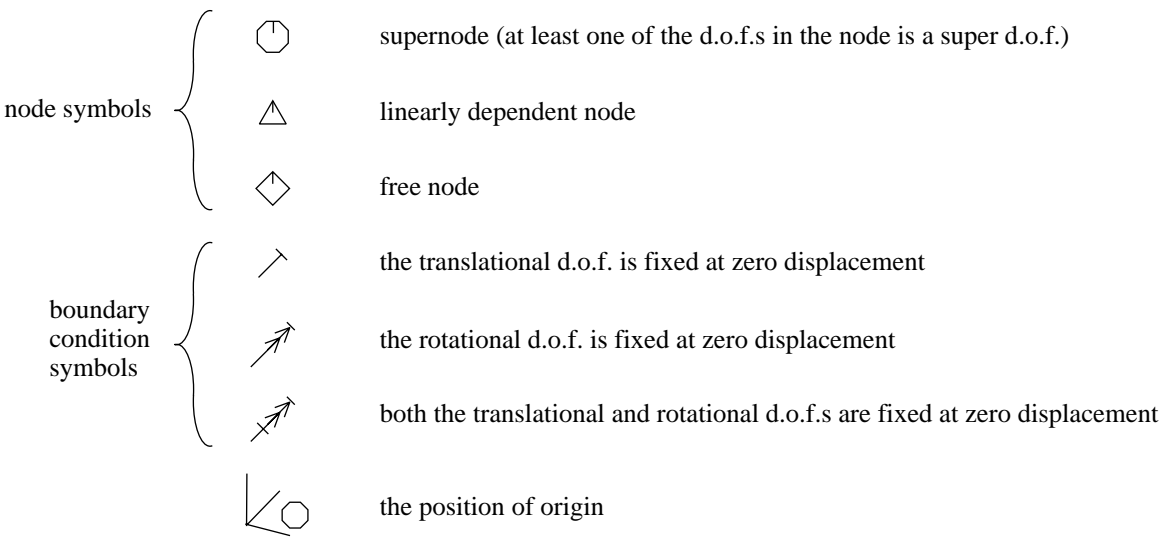


Figure 5.12 Symbols produced by the LABEL command

LABEL CONCEPT-ATTRIBUTES

...	CONCEPT-ATTRIBUTES	HYDRODYNAMIC	DRAG-COEFF		...	LOCAL-X-AXIS	
			INERTIA-COEFF			LOCAL-Y-AXIS	
			FLOODING-COEFF			LOCAL-Z-AXIS	
			STABILITY	BUCKLING-FACTOR		...	LOCAL-Y-AXIS
		BUCKLING-LENGTH		LOCAL-Z-AXIS			
		PILE-DATA	FIXED-TO				
			GAMMA-FLUID				
			TIP-CODE				
			YIELD-STRENGTH				

PURPOSE:

This command labels the concept attribute values connected to the displayed member concepts.

PARAMETERS:

HYDRODYNAMIC	Add hydrodynamic attributes assigned to concepts.
DRAG-COEFF	Add drag coefficient value.
INERTIA-COEFF	Add inertia coefficient value.
FLOODING-COEFF	Add flooding coefficient value.
STABILITY	Add stability parameters assigned to concepts.
BUCKLING-FACTOR	Add buckling factor value.
BUCKLING-LENGTH	Add buckling length value.
LOCAL-X-AXIS	Add value assigned to local x axis.
LOCAL-Y-AXIS	Add value assigned to local y axis.
LOCAL-Z-AXIS	Add value assigned to local z axis.
PILE-DATA	Add the concept attribute values connected to the displayed pile concepts.
FIXED-TO	Label the pile group fixed to node reference.
GAMMA-FLUID	Label the unit weight of fluid / soil inside the pile.

Label the pile material yield strength.

```
SET GRAPHICS SIZE-SYMBOLS ...
```

LABEL SOIL-DATA

...	SOIL-DATA	SOIL-TYPE	soil-id
		Z-LEVEL	

PURPOSE:

The command adds labels on the display of the soil profile.

PARAMETERS:

SOIL-TYPE	Adds the soil types, i.e. SAND / CLAY + soil number.
Z-LEVEL	Z levels at bottom of each soil division (type).
soil-id	The soil profile id (number) to be used. Currently only id 1 allowed.

NOTES:

The command should preferably be used after the ADD-DISPLAY SOIL-PROFILE command.

The size of the symbols equals the size used for the element-numbers (SET GRAPHICS SIZE-SYMBOLS ELEMENT-NUMBERS value).

See also:

SET GRAPHICS SIZE-SYMBOLS ...

LINEAR-DEPENDENCY

LINEAR-DEPENDENCY	GENERAL-NODE-DEPENDENCY	...
	TWO-NODE-DEPENDENCY	
	END	

PURPOSE:

The command defines the displacements of selected nodes to be linearly dependent of displacements of other selected nodes. See also Section 3.12.

The GENERAL-NODE-DEPENDENCY option may couple any d.o.f. of a node (the dependent d.o.f.) to any other d.o.f.s of any other nodes (the independent d.o.f.s). The TWO-NODE-DEPENDENCY option is used to couple all d.o.f.s of a given node to the corresponding d.o.f.s of two other nodes.

Linear dependencies involves that the dependent d.o.f.s get the boundary condition LINEAR and the independent d.o.f.s get the boundary condition SUPERL (SUPER due to linear dependency). (A SUPERL d.o.f. will appear in the next level superelement in the same way as a SUPER d.o.f.) There are certain rules as concerns the boundary condition of a d.o.f. before and after the definition of a linear dependency and whether the linear dependency can at all be defined. Table 5.2 describes these rules for the dependent d.o.f. and Table 5.3 for the independent d.o.f. A violation of the rules involves that the linear dependency is not accepted.

Table 5.2 Rules for boundary condition of a dependent d.o.f.

Boundary condition before	Boundary condition after	Comment
FREE.	LINEAR	OK
FIXED	LINEAR	Warning: The boundary condition is changed.
PRESC	-	Illegal
LINEAR	LINEAR	OK (implies adding dependency of new d.o.f.s)
SUPER	LINEAR	Warning: Boundary condition is changed.
SUPERL	-	Illegal. Linear dependency cannot propagate.

Table 5.3 Rules for boundary condition of an independent d.o.f.

Boundary condition before	Boundary condition after	Comment
FREE	SUPERL	OK if FORCE-INTO-SUPER
FIXED	SUPERL	OK if FORCE-INTO-SUPER
PRESC	-	Illegal, cannot be changed to super.

Table 5.3 Rules for boundary condition of an independent d.o.f.

LINEAR	-	Illegal, d.o.f. is not independent.
SUPER	SUPERL	OK
SUPERL	SUPERL	OK

LINEAR-DEPENDENCY GENERAL-NODE-DEPENDENCY

...	GENERAL-NODE-DEPENDENCY						...
...	{dep-node	dep-dof	{indep-node	indep-dof	beta	}*	}*

PURPOSE:

The command defines general linear dependency between nodes. See also Section 3.12.

The dependency is defined by selecting a single d.o.f. of a node to be dependent of any other d.o.f.s. To define the other d.o.f.s of the same node also to be dependent the command must be re-entered.

PARAMETERS:

dep-node	Node number of the dependent node. (Slave.)
dep-dof	D.o.f. to be dependent, legal specifications are: X — Translation in x-direction Y — Translation in y-direction Z — Translation in z-direction R-X — Rotation about the x-direction R-Y — Rotation about the y-direction R-Z — Rotation about the z-direction END
indep-node	Node number of an independent node. (Master.)
indep-dof	The independent d.o.f. to, legal specifications are: X — Translation in x-direction Y — Translation in y-direction Z — Translation in z-direction R-X — Rotation about the x-direction R-Y — Rotation about the y-direction R-Z — Rotation about the z-direction FORCE-X-INTO-SUPER

FORCE-Y-INTO-SUPER
FORCE-Z-INTO-SUPER
FORCE-R-X-INTO-SUPER
FORCE-R-Y-INTO-SUPER
FORCE-R-Z-INTO-SUPER
END

If the independent d.o.f. has not previously been defined as SUPERL it will be defined as SUPERL by giving these alternatives

beta

Linear dependency factor.

LINEAR-DEPENDENCY TWO-NODE-DEPENDENCY

...	TWO-NODE-DEPENDENCY	...
...	{dep-node	indep-node1
	FORCE-INTO-SUPER	indep-node1
...	indep-node2	beta
	FORCE-INTO-SUPER	indep-node2
		}*

PURPOSE:

The command defines linear dependency of a node on two other nodes. All d.o.f.s of the dependent node are dependent of the corresponding d.o.f.s of the first independent node by the factor beta and the second independent node by the factor (1 - beta). See also Section 3.12.

PARAMETERS:

dep-node	Node number of the dependent node. (Slave.)
indep-node1	Node number of the first independent node.
FORCE-INTO-SUPER	Using this option the d.o.f.s of the independent nodes can be forced into SUPERL if they are not SUPER or SUPERL already.
indep-node2	Node number of the second independent node. (Master.)
beta	Linear dependency factor.

LOAD

LOAD	load-case	ELEMENT	CONSTANT-TEMPERATURE-ACROSS-THICKNESS		...
			DISTRIBUTED		
			LINE-LOAD		
			POINT		
		GRAVITY			
		NODE	FORCE		
			PRESCRIBED-ACCELERATION		
			PRESCRIBED-DISPLACEMENT		
		ROTATION-OF-STRUCTURE			

PURPOSE:

The command defines loads.

Loads can be changed by the CHANGE LOAD command, and deleted by the DELETE LOAD command.

The following types of loads can be defined:

- Nodal loads:
 - Nodal forces and moments — The load may be given for any boundary condition code (FREE, FIXED, PRESCRIBED, SUPER).
 - Nodal prescribed displacements or accelerations — In effect, loads are applied corresponding to the given displacements or accelerations. To specify this kind of loading the boundary condition code PRESCRIBED must be defined for the appropriate nodes (d.o.f.s). See the BOUNDARY command.
- Element loads:
 - Element point load — The given forces are applied at a point along the element defined by a given distance from end 1 of the element. This can only be assigned to BEAM type elements.
 - Element distributed load — The load is distributed along (a part of) the element. The load intensity is given at two points along the element. The points are defined by their distances from the two ends of the element. The load varies linearly along the element between the two points.
 - Element line load — The load is distributed along several element. The load intensity is given at start and end of chain of elements.
 - Element temperature load — Temperatures can be assigned to TRUSS and BEAM elements. Two temperature load options are available:
 - Same temperature for all (the two) nodes of the element
 - Different temperature for all (the two) nodes of the element

- Gravity load — The acceleration of gravity is given, the analysis program (e.g. Sestra) will compute the weight distribution for the model accounting for any nodal masses and element masses. The element masses are based on:
 - Density of elements defined by the PROPERTY MATERIAL command
 - Cross sectional area of elements defined by the PROPERTY SECTION command
 - Length of the elements being the node-to-node distance or the element end-to-end distance; see the detailed description of the command.
- Rotation of structure — An acceleration field due to angular velocity (a centripetal acceleration field) and/or acceleration (tangential acceleration field) about an arbitrary axis is put up. The resulting inertia loads are computed by the analysis program in the same manner as described for the gravity load above.

The loads should be given consecutive load case numbers (1, 2, 3, ...) or else computation time will be spent in the analysis program (Sestra) solving zero load cases.

A single load case may contain all and any of the load types above. And except for the gravity and 'rotation of structure' loads, the same load type may be repeated several times.

Several loads may be specified for the same node/element within the same load case. A load index is used to distinguish between individual loads of the same type for the same node/element for the same load case. For example, a nodal force defined for the second time for the same node for the same load case is given index 2. Note that the load index may change after deletion; the load index always goes from 1 to N where N is the number of loads of the same type for that particular node/element. This load index has to be specified when loads are changed or deleted. The load index is shown in the printout of the load data.

Loads may be specified as real loads, real and imaginary loads, or as loads having amplitudes and a phase angles (in degrees).

LOAD load-case ELEMENT CONSTANT-TEMPERATURE-ACROSS-THICKNESS

...	load-case	ELEMENT	CONSTANT-TEMPERATURE-ACROSS-THICKNESS				...
...	SAME-FOR-ALL-NODES		select-elements	temp			
	DIFFERENT-FOR-ALL-NODES		select-elements	temp1	temp2		
	END						

PURPOSE:

The command defines element temperature loads for selected elements. The values are specified at two nodes of the element or as the same value at both nodes.

PARAMETERS:

load-case	Load case number
select-elements	Select elements; see Section 5.1.
temp	Temperature at both nodes.
temp1	Temperature at node 1.
temp2	Temperature at node 2.

LOAD load-case ELEMENT DISTRIBUTED

...	load-case	ELEMENT	DISTRIBUTED		select-elements	...	
...	{	GLOBAL		fxj	fyj	fzj	...
		LOCAL					
		TRANSFORMATION	trano				

...	END				dj	}*2
	IMAGINARY-COMPLEX	ifxj	ifyj	ifzj		
	PHASE-COMPLEX	pxfj	pfyj	pfzj		

PURPOSE:

The command defines loads distributed along (parts of) selected elements. The load intensities are given at two points along the elements. The points are defined by their distances from the two ends of the element. The loads vary linearly along the elements between these two points.

The command SET ELEMENT-LOAD-DISTANCE-MODE may be used to specify that the distances to the points (where the load intensities are given) are from the ends of the flexible part of the element (relevant when eccentricities are defined) rather than from the projection of the nodes onto the element axis (the default).

PARAMETERS:

load-case	Load case number.
select-elements	Select elements; see Section 5.1.
trano	Transformation reference number.
fxj fyj fzj	Real components of the force intensity at a point of distance dj from end j.
ifxj ifyj ifzj	The corresponding imaginary components.
pxfj pfyj pfzj	The corresponding phase angle components in degrees.
	The real components are treated as amplitudes.
dj	Distance from end j.

NOTES:

Warning: If the element load is defined in the local (i.e. the element's) coordinate system then do not change this coordinate system by the PROPERTY LOCAL-COORDINATE command as such will not correspondingly change the load. If required, use PROPERTY LOCAL-COORDINATE prior to the load definition.

The user may select between **END**, **IMAGINARY-COMPLEX** and **PHASE-COMPLEX** for the first end only.

LOAD load-case ELEMENT LINE-LOAD

...	load-case	ELEMENT	LINE-LOAD	node1	node2	...	
...	{	GLOBAL		fxj	fyj	fzj	...
		LOCAL					
		TRANSFORMATION	trano				
...	END				}*2		
	IMAGINARY-COMPLEX	ifxj	ifyj	ifzj			
	PHASE-COMPLEX	pfxj	pfyj	pfzj			

PURPOSE:

The command defines loads distributed along line of selected elements. The load intensities are given at start node and end node of chain of elements.

PARAMETERS:

load-case	Load case number.
node1	Start node
node2	End node
trano	Transformation reference number.
fxj fyj fzj	Real components of the force intensity at a point of distance dj from end j.
ifxj ifyj ifzj	The corresponding imaginary components.
pxj pyj pfzj	The corresponding phase angle components in degrees.
	The real components are treated as amplitudes.

NOTES:

Warning: If the element load is defined in the local (i.e. the element's) coordinate system then do not change this coordinate system by the PROPERTY LOCAL-COORDINATE command as such will not correspondingly change the load. If required, use PROPERTY LOCAL-COORDINATE prior to the load definition.

The resulting loads are implemented as distributed loads on each of the elements along the selected line.

The user may select between END, IMAGINARY-COMPLEX and PHASE-COMPLEX for the first end only.

LOAD load-case ELEMENT POINT

...	load-case	ELEMENT	POINT	select-elements	...	
...	GLOBAL		fx	fy	fz	...
	LOCAL					
	TRANSFORMATION	trano				

...	END				d	
	IMAGINARY-COMPLEX		ifx	ify		ifz
	PHASE-COMPLEX		pfx	pfy		pfz

PURPOSE:

The command defines element point loads.

Element point loads may only be given at a distance from the element ends of at least 1/200 of the element length. (The reason for this restriction is that an element point load is converted to an element distributed load acting over 1/100 of the element length.)

The command SET ELEMENT-LOAD-DISTANCE-MODE may be used to specify that the distance to the point (where the load is given) is from end 1 of the flexible part of the element (relevant when eccentricities are defined) rather than from the projection of node 1 onto the element axis (the default).

PARAMETERS:

load-case	Load case number.
select-elements	Select elements; see Section 5.1.
tran	Transformation reference number.
fxj fyj fjz	Real components of the force intensity at a point of distance d from end 1.
ifxj ifyj ifzj	The corresponding imaginary components.
pfxj pfyj pfzj	The corresponding phase angle components in degrees.
	The real components are treated as amplitudes.
d	Distance from end 1.

NOTES:

Warning: If the element load is defined in the local (i.e. the element's) coordinate system then do not change this coordinate system by the PROPERTY LOCAL-COORDINATE command as such will not correspondingly change the load. If required, use PROPERTY LOCAL-COORDINATE prior to the load definition.

LOAD load-case GRAVITY

...	load-case	GRAVITY	YES	gx	gy	gz
			NO			

PURPOSE:

The command defines the acceleration of gravity used (by the analysis program) to compute the weight of elements and nodal masses.

PARAMETERS:

load-case Load case number.

YES Answer 'yes' to question: Weight contribution from flexible part of elements only?
This option has significance only if eccentricities (offsets) are defined.

NO Answer 'no' to question above. Possible eccentricities are neglected and the contribution to the weight is for each element based on the distance between the two nodes.

gx gy gz Components of acceleration of gravity in the directions of the global axes. An acceleration of gravity will yield a gravity force in the same direction as the acceleration.

LOAD load-case NODE FORCE

...	load-case	NODE	FORCE	select-nodes	...					
...	GLOBAL			fx	fy	fz	mx	my	mz	...
	TRANSFORMATION		trano							
...	END									
	IMAGINARY-COMPLEX			ifx	ify	ifz	imx	imy	imz	
	PHASE-COMPLEX			pfx	pfy	pfz	pmx	pmy	pmz	

PURPOSE:

The command defines nodal forces and moments.

PARAMETERS:

load-case	Load case number.
select-nodes	Select nodes; see Section 5.1.
trano	Transformation reference number.
fx fy fz mx my mz	Real components of forces and moments.
ifx ify ifz imx imy imz	Imaginary components of forces and moments for a complex load.
pfx pfy pfz pmx pmy pmz	The corresponding phase angle components in degrees.
	The real components are treated as amplitudes.

LOAD load-case NODE PRESCRIBED-ACCELERATION/-DISPLACEMENT

...	load-case	NODE	PRESCRIBED-ACCELERATION					select-nodes	...
			PRESCRIBED-DISPLACEMENT						
...	GLOBAL		tx	ty	tz	rx	ry	rz	...
	TRANSFORMATION	trano							
...	END								
	IMAGINARY-COMPLEX		itx	ity	itz	irx	iry	irz	
	PHASE-COMPLEX		ptx	pty	ptz	prx	pry	prz	

PURPOSE:

The command defines nodal prescribed displacements or accelerations for selected nodes. The nodes must have been given the boundary condition code PRESCRIBED.

Alternatively to all six, only selected d.o.f.s may be given prescribed displacements/accelerations. The boundary condition code PRESCRIBED must then have been defined only for the relevant d.o.f.s. Note that even in this case, a value must be entered for all six d.o.f.s by giving the value 0.0 for the non-prescribed d.o.f.s.

PARAMETERS:

load-case	Load case number.
select-nodes	Select nodes; see Section 5.1.
trano	Transformation reference number.
tx ty tz rx ry rz	Real displacement/acceleration components for the translational and rotational d.o.f.s. Rotational d.o.f.s are given in radians.
itx ity itz irx iry irz	Imaginary displacement/acceleration components for a complex load. Rotational d.o.f.s are given in radians.
ptx pty ptz prx pry prz	The corresponding phase angle components in degrees. The real components are treated as amplitudes.

LOAD load-case ROTATION-OF-STRUCTURE

...	load-case	ROTATION-OF-STRUCTURE					...	
...	p1x	ply	p1z	p2x	p2y	p2z	ang-vel	ang-acc

PURPOSE:

The command defines an acceleration field composed of centripetal acceleration due to angular velocity, and tangential acceleration due to angular acceleration about an arbitrary axis.

The direction of rotation is according to the right hand rule with respect to the direction point 1 ® point 2 (the thumb points in this direction). This direction of rotation has no consequence for the centripetal acceleration (angular velocity).

Note that the angular acceleration is a forced rotation and not a rotational acceleration field. This means that the inertia forces due to the angular acceleration have the opposite direction of the direction of rotation. (This may be compared with giving the support points of a structure the acceleration g upwards instead of introducing an acceleration of gravity field acting downwards. In both cases the inertia forces will act downwards.)

Note that the angular velocity and acceleration are given in radians.

PARAMETERS:

load-case	Load case number.
p1x ply p1z	Point 1 for defining the axis of rotation.
p2x p2y p2z	Point 2 for defining the axis of rotation.
ang-vel	Angular velocity in radians/time-unit
ang-acc	Angular acceleration in radians/(time-unit) ²

MASS-ON-NODE

MASS-ON-NODE	select-nodes	mass-tx	mass-ty	mass-tz	mass-rx	mass-ry	mass-rz
--------------	--------------	---------	---------	---------	---------	---------	---------

PURPOSE:

The command defines nodal masses. This will be a diagonal mass matrix added to any mass contribution from the elements and from added mass.

While the unit of the translational masses is mass (kg, tons, etc. according to the chosen set of consistent units), the unit of the rotational masses will be mass*length² (kg*mm², tons*m², etc.). In comparison with the translational masses, the rotational masses will normally contribute little to the solution and may be given the value zero.

Having defined a nodal mass, a new MASS-ON-NODE command for the same node will accumulate masses (a proper warning will be given).

PARAMETERS:

select-nodes Select nodes; see Section 5.1.

mass-tx mass-ty mass-tz Masses with respect to translational d.o.f.s

mass-rx mass-ry mass-rz Masses with respect to the rotational d.o.f.s

NOTES:

See also:

PRINT NODE MASS-ON-NODE ...

NODE

NODE	nodeno	...
	EXTRAPOLATION	
	GROUP	
	INTERSECTION	
	LINE	
	RELATIVE	
	END	

PURPOSE:

The command creates nodes in different ways:

- a single node defined by its number and coordinates,
- a single node extrapolated (shot out) along a line through two existing nodes,
- a group of nodes defined relative to previously created nodes,
- a single node at the point of intersection of two straight lines through existing nodes,
- a line of nodes distributed along a straight line segment between two previously created nodes, and
- a single node positioned relative (by offset) to an existing node.

Existing nodes can be given new positions (coordinates) by the CHANGE NODE command. The node numbers can be changed by the RENUMBER command. Created nodes can be deleted by the DELETE command.

See Section 2.5 for an explanation of the aspects and consequences of the node numbering.

NOTES:

Node numbers are limited to seven digits.

NODE EXTRAPOLATION

EXTRAPOLATION	node1	node2	DISTANCE	dist	nodeno	
			XY-PLANE-INTERSECTION		z-value	nodeno
			YZ-PLANE-INTERSECTION		x-value	nodeno
			ZX-PLANE-INTERSECTION		y-value	nodeno

PURPOSE:

The command creates a node extrapolated (shot out) along a line through two existing nodes. Note that the new node may also fall between the two existing nodes in which case an interpolation is performed.

PARAMETERS:

node1 node2	First and second node defining the line.
DISTANCE	Position the node at a given distance from node2.
dist	The distance from node2 to nodeno. A positive value is measured in the direction away from node1.
nodeno	Number of the node to create.
XY-PLANE-INTERSECTION	Position the node at the intersection between the line and the given XY-plane.
z-value	Z-coordinate defining the XY-plane.
YZ-PLANE-INTERSECTION	Position the node at the intersection between the line and the given YZ-plane.
x-value	X-coordinate defining the YZ-plane.
ZX-PLANE-INTERSECTION	Position the node at the intersection between the line and the given ZX-plane.
y-value	Y-coordinate defining the ZX-plane.

EXAMPLES:

```

NODE EXTRAPOLATION 22 24 DISTANCE 5 27
                   21 23 YZ-PLANE-INTERSECTION 10 26

```

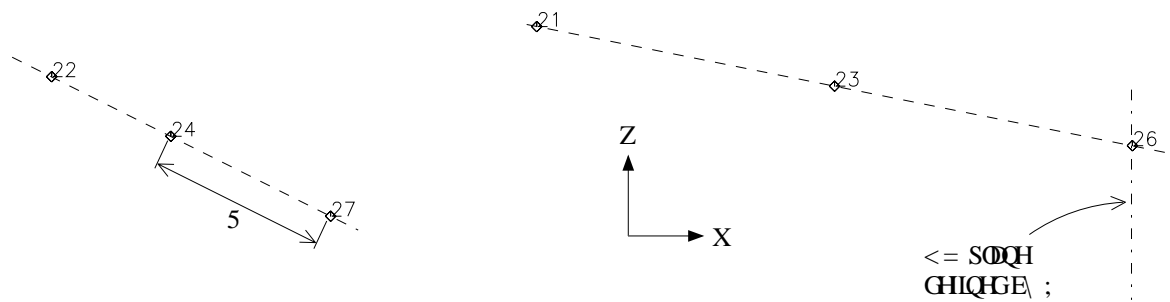


Figure 5.13 Creating a node by NODE EXTRAPOLATION

NODE GROUP

...	GROUP	node1	node2	nstep	refnod	refstep	dx	dy	dz
-----	-------	-------	-------	-------	--------	---------	----	----	----

PURPOSE:

The command creates a group of nodes relative to previously created nodes.

PARAMETERS:

node1	Node number of the first node to be created.
node2	Node number of the last node to be created.
nstep	Step in the node numbering of nodes to be created.
refnod	First reference node, i.e. the coordinates of node1 will be defined relative to refnod.
refstep	Step in the reference node numbering, i.e. the coordinates of node node1+nstep will be defined relative to node refnod+refstep. All reference nodes must previously have been created.
dx dy dz	The relative position of the new nodes with respect to the corresponding reference node. The coordinates must be specified in the global coordinate system.

EXAMPLES:

```
NODE GROUP 21 26 1 11 1 0. 10. 0.
```

See Figure 5.14; the new nodes 21 through 26 are created based on the existing group of nodes 11 through 16. The node numbering step is 1 for both the new nodes and the reference nodes. The relative positions of the new nodes with respect to the corresponding reference nodes are given as the vector (0,10,0).

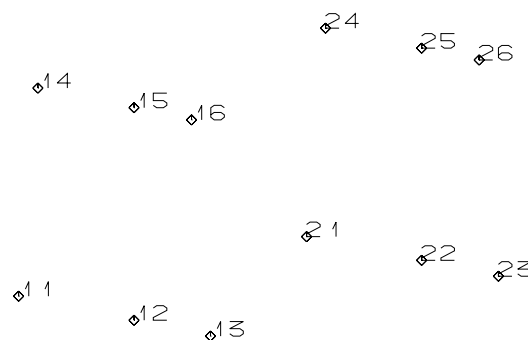


Figure 5.14 Creating a NODE GROUP

NODE INTERSECTION

...	INTERSECTION	node1	node2	node3	node4	nodeno
-----	--------------	-------	-------	-------	-------	--------

PURPOSE:

The command creates a node at the point of intersection of two straight lines. Each line is defined by two nodes that previously must have been created.

PARAMETERS:

- node1 node2
 First and second node defining the first line.
- node3 node4
 First and second node defining the second line.
- nodeno
 Number of the node to create.

EXAMPLES:

```

NODE INTERSECTION 11 15 12 14 13
                  21 22 23 24 25

```

See Figure 5.15; the new node 13 is created at the point of intersection of the two lines between nodes 11 and 15, and 12 and 14 respectively. In the other example the new node (25) is located at the extrapolation of the two lines.

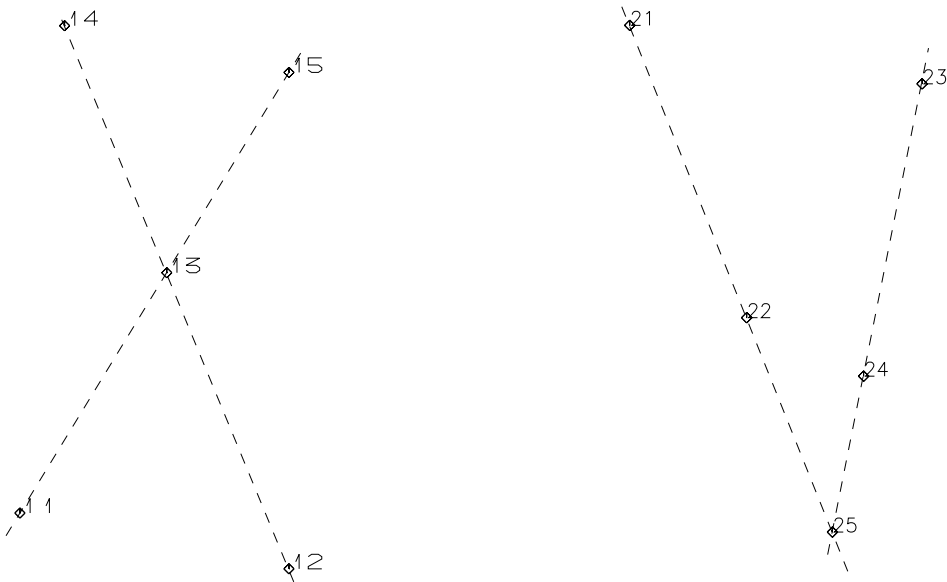


Figure 5.15 Creating a node by NODE INTERSECTION

NODE LINE

...	LINE	node1	node2	ndiv	nodeno	*		EVEN	
					STEP	startnode	nstep	space	*
					AUTO				

PURPOSE:

The command creates a line of nodes distributed along a straight line segment defined by two previously created nodes.

PARAMETERS:

node1 node2	The two existing nodes defining the line segment. The direction of the line is from node1 to node2.
ndiv	Number of divisions of the line segment. ndiv - 1 number of nodes will be created.
nodeno*	Node numbers of the created nodes. The user manually assigns a number to each node.
STEP	Node numbers will be generated stepwise.
startnode	Number of the first created node.
nstep	The step in the node numbering.
AUTO	Automatic node numbering. The program will generate the node numbers sequentially, starting with the highest current node number plus one.
EVEN	The line will be divided into ndiv equal parts.
space*	The spacings between the nodes starting in node1. If ndiv spacings are entered they will be interpreted as relative spacings. If less than ndiv spacings, they are interpreted as true spacings and the remaining part of the line will be divided into equal parts.

EXAMPLES:

```
NODE  LINE  11  61  5  STEP  21  10  1.0  0.9  0.8  0.7  0.5
```

See Figure 5.16; the new nodes 21 through 51 are created along the line between the previously created two nodes 11 and 61. There are 5 divisions for which relative spacings are given.

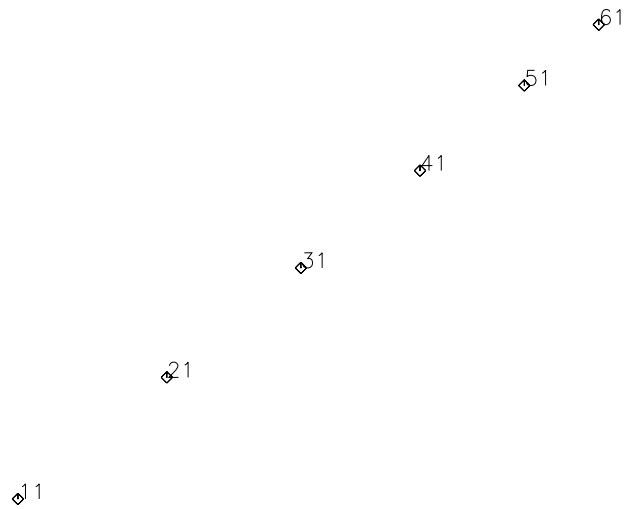


Figure 5.16 Creating a NODE LINE

NODE RELATIVE

...	RELATIVE	refnode	dx	dy	dz	nodeno
-----	----------	---------	----	----	----	--------

PURPOSE:

The command creates a node positioned relative (by offset) to an existing node.

PARAMETERS:

refnode Reference node from which the new node is set off.
dx dy dz Offset given in the global coordinates of the model.
nodeno Number of the node to create.

EXAMPLES:

```
NODE RELATIVE 21 5 0 -1 23
```

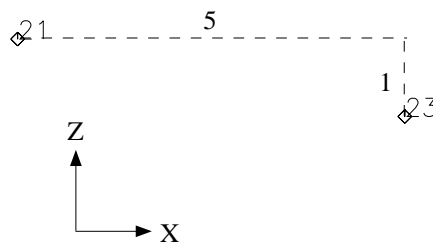


Figure 5.17 Creating a node by NODE RELATIVE

NODE single

...	nodeno	x	y	z
-----	--------	---	---	---

PURPOSE:

The command creates a new single node.

PARAMETERS:

nodeno	Node number of the new node.
x y z	Cartesian coordinates of the node.

PLOT

PLOT	AS-LAST-DISPLAY	...	YES	...	YES	...	SUPER-NODES-ONLY	...
	ALL		NO		NO		ALL-NODES	
	NONE						NONE	

...	EXTERNAL-NODE-NUMBER	...	ELEMENT-NUMBER	...	LOCAL-Y-AXIS	...
	INTERNAL-NODE-NUMBER		SECTION-NUMBER		LOCAL-Z-AXIS	
	NONE		MATERIAL-NUMBER		NONE	
			MEMBER-NAMES			
			CONCEPT-ATTRIBUTES			
			PILE-NAMES			
			SOIL			
			NONE			

...	4*	textline	A1
			A2
			A3
			A4
			A5
			OTHER plot-width

PURPOSE:

The command generates a plot file of the last display (AS-LAST-DISPLAY) or the complete model (ALL). How to send the plot file to an off-line plotter or laser printer varies with the installation; see the SESAM System Manual /1/. Figure 5.18 shows the appearance of a plot.

The user decides whether origin symbol (the first YES/NO), boundary condition symbols (the second YES/NO), node symbols, node numbers, element/section/material numbers or member/pile names or concept attribute values, and local coordinate symbols are wanted. Four lines of text, maximum 24 characters each, are reproduced on the plot. Finally the plot format (A1/A2/...) is chosen.

Note that possible labelling (LABEL command) of the screen display will not appear on the plot. This is instead achieved by the PLOT command as outlined above. The ROTATE and SET GRAPHICS commands, however, will have impact on the plot.

Additional subselections to SOIL regarding SOIL-TYPE / Z-LEVEL as described for LABEL SOIL-DATA.

Additional sub-selections to CONCEPT-ATTRIBUTES regarding HYDRODYNAMIC / STABILITY / PILE-DATA as described for LABEL CONCEPT-ATTRIBUTES.

The date and time when the plot was generated will be given on the plot together with the scale, the superelement number and the superelement level (=1). The scale shown on the plot will be correct only if the coordinates are given in metres.

Note that text lines containing blank characters must be enclosed in apostrophes: 'this is an example text'.

PARAMETERS:

textline Text line

plot-width Width of plot in metres. Height of plot is 1.4 times width of plot.

NOTES:

The SOIL option should preferably be used after the ADD-DISPLAY SOIL-PROFILE command.

See also:

SET GRAPHICS PLOT-FILE ...

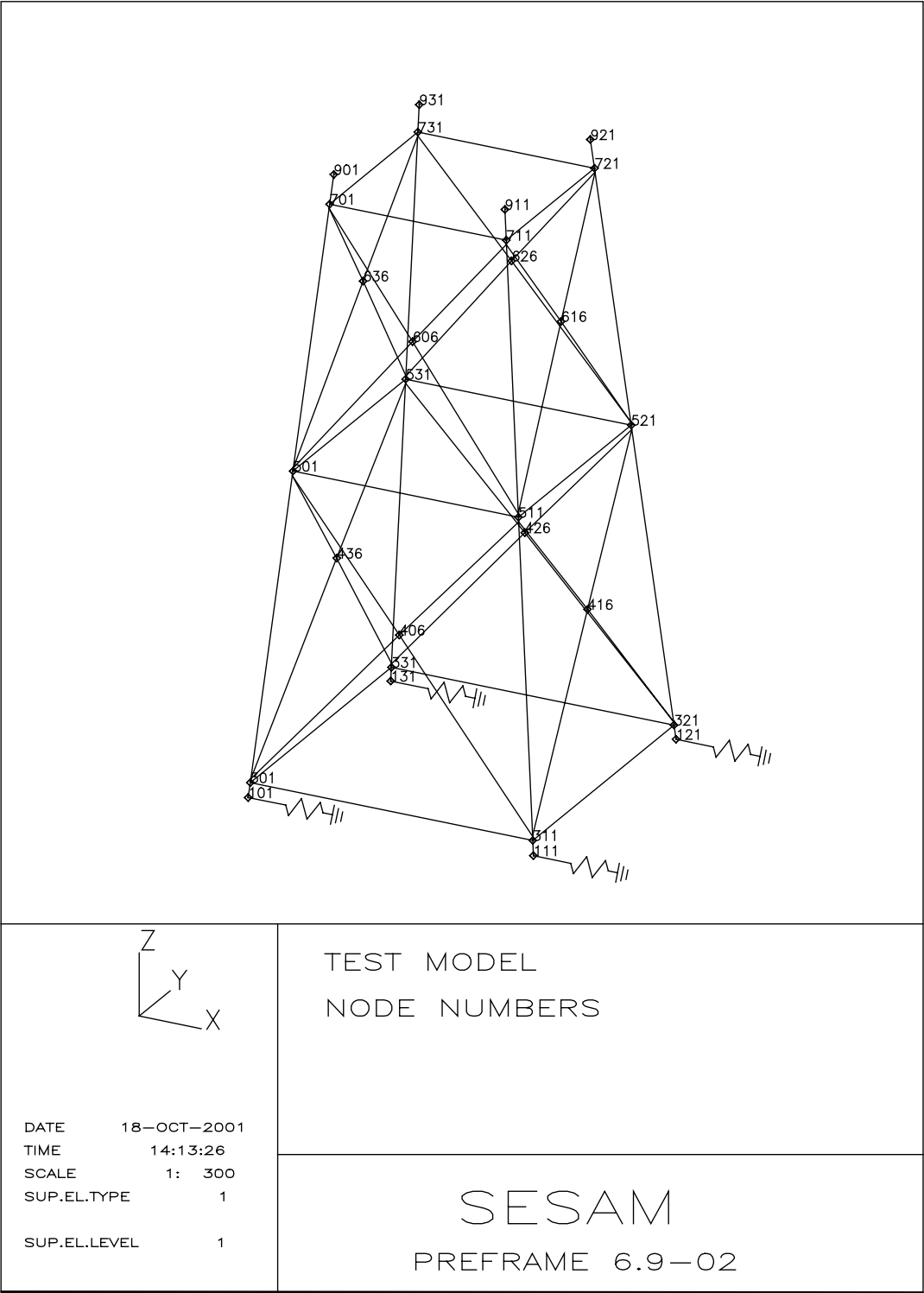


Figure 5.18 Plot of model

PRINT

PRINT	ALL	sub-commands
	CONCEPT-ATTRIBUTES	
	DATA-CHECK	
	ECCENTRICITY	
	ELEMENT	
	HINGE	
	LOADS	
	LOCAL-COORDINATE	
	MASS-OF-ELEMENTS	
	MATERIAL	
	NODE	
	SECTION	
	SOIL	
	STATUS	
	STUCTION-CONCEPT	
	TRANSFORMATION	

PURPOSE:

The command prints data in tables either on the screen or to a file. The destination is controlled by the SET PRINT DESTINATION command. The SET PRINT FILE command may be used to specify the name of the print file which by default will be the name of the model file with file extension LIS.

By default, PRINT ALL will go to a file while all other PRINT commands will go to the screen. The ALL option prints a complete set of tables for the model, i.e. all data (except concepts with attributes) available for tabulation will be printed.

Real values are normally printed in F format for best readability. Values too large to be printed within the given field will be indicated as such by asterisks (****). The user may decide to use FORTRAN E or G format by using the SET PRINT FORMAT command.

NOTES:

See also:

SET PRINT ...

PRINT CONCEPT-ATTRIBUTES

...	CONCEPT-ATTRIBUTES	HYDRODYNAMIC
		STABILITY
		PILE-DATA

PURPOSE:

The command prints a table of structural concept attributes (hydrodynamic, stability or pile parameters) connected to the concepts. The table has the following appearance:

Example of print member data

SUPER ELEMENT TYPE: 1 LEVEL: 1

--- CONCEPT ---			--ATTRIBUTES--					
NO.	NAME	Type	Attrib	Value	Attrib	Value	Attrib	Value
1	301422	HYDRO	Floo =	1.0000				
		HYDRO	CDy =	0.7000	CDz =	0.7000		
		HYDRO	CMy =	2.0000	CMz =	2.0000		
6	301122	HYDRO	Floo =	1.0000				
		HYDRO	CDy =	0.7000	CDz =	0.7000		
		HYDRO	CMy =	2.0000	CMz =	2.0000		
		CODE	BLen =	L-B-J				
6	301122	CODE	BFKy =	0.8000	BFKz =	0.8000		
		CODE	BLy =	0.1200E+02	BLz =	0.1300E+02		

Example of print pile data

SUPER ELEMENT TYPE: 1 LEVEL: 1

--- CONCEPT ---			--ATTRIBUTES--					
NO.	NAME	Type	Attrib	Value	Attrib	Value	Attrib	Value
43	P71	PILE	Tipcod	1				
		MATER.	Fy =	0.3450E+09				
51	P11	PILE	Fix to	100				
		PILE	Tipcod	1				
		MATER.	Fy =	0.3450E+09				
55		SECT.	EA-new	0.1111E+04	EI-new	0.2222E+04	GA-new	0.3333E+04
			GT-new	0.4444E+04				
58	P81	PILE	Fix to	100				
		PILE	Tipcod	1				
		MATER.	Fy =	0.3450E+09				
62		SECT.	EA-new	0.1111E+04	EI-new	0.2222E+04	GA-new	0.3333E+04
			GT-new	0.4444E+04				

PARAMETERS:

HYDRODYNAMIC

Print hydrodynamic attributes assigned to concepts.

STABILITY

Print stability parameters assigned to concepts.

PILE-DATA

Print pile parameters assigned to concepts.

NOTES:

See also:

LABEL CONCEPT-ATTRIBUTES ...

PRINT DATA-CHECK

...	DATA-CHECK	BEAM-ELEMENTS
		NODES

PURPOSE:

The command prints check data for verification of the model.

PARAMETERS:

BEAM-ELEMENTS	Print a list of beam elements with missing material and/or cross section properties.
NODES	Print a list of nodes not connected to any elements.

PRINT ECCENTRICITY

...	ECCENTRICITY	select-elements
-----	--------------	-----------------

PURPOSE:

The command prints a table of eccentricities (offsets) defined for the elements. The table has the following appearance:

```

SUPER ELEMENT TYPE:      1  LEVEL:      1
OFFSETS IN SUPERELEMENT'S COORDINATE SYSTEM, FROM NODE TO ELEMENT
EXT.    INT.    ECCENTRICITIES AT ODD NODES    ECCENTRICITIES AT EVEN NODES
EL.      EL.      X        Y        Z        X        Y        Z
-----
   324    112      0.495   -0.495   -4.951     0.000   -3.000    0.000
   325    116     -2.062    0.767    3.027    -0.090    2.738    2.549
   424    113      0.495   -0.495   -4.951     0.000   -3.000    0.000
   524    114      0.297   -0.297   -2.970     0.000   -5.000    0.000

```

columns of the table are, from left to right:

- user defined (external) element number,
- internal element number (the first element created is number 1; the last is number NEL where NEL is the number of basic elements), this number is normally of no interest to the user,
- eccentricity at node 1 ('odd node') of the element given as a vector in the global coordinate system,
- eccentricity at node 2 ('even node') of the element given as a vector in the global coordinate system,
--eccentricities in additional nodes for elements with more than two nodes are given on succeeding lines.

PARAMETERS:

select-elements Select elements; see Section 5.1.

PRINT ELEMENT

...	ELEMENT	select-elements
-----	---------	-----------------

PURPOSE:

The command prints a table of the basic elements. The table has the following appearance:

SUPER ELEMENT TYPE:			1	LEVEL:			1			
EXT.	INT.	EL.	MAT.	SECT.	SECT.	SECT.	ELEMENT LENGTH			
EL.	EL.	TYPE	NO.	NO.	TYPE	D H TH	FLEXIBLE PART	NODE 1	NODE 2	
101	1	BEAS	4	2	CHAN.	4.00	5.459192	101	201	
111	9	BEAS	4	4	BOX	7.00	5.459198	105	205	
234	130	AXIS	6				29.791088	304	206	
235	131	AXDA	7				29.791090	308	206	
732	57	TESS	4	11	PIPE	12.00	10.000000	704	706	
1020	15	PILS	99	99	GEN.			105		
9004	127	GSPR	2					104		
9005	129	GDAM	55					105		
9008	128	GSPR	3					108		

columns of the table give, from left to right:

- user defined (external) element number,
- internal element number (the first element created is number 1; the last is number NEL where NEL is the number of basic elements), this number is normally of no interest to the user,
- basic element type; see Section 5.1 for an overview of the element types that Preframe may create; other element types with other names may have been created by other preprocessors and read into Preframe by the READ command,
- number of material connected to the element,
- number of cross section connected to the element,
- type of cross section connected to the element,
- ‘primary’ cross section parameter (diameter, height or thickness),
- flexible element length (refer to possible eccentricities) if two node element,
- user defined (external) numbers of the nodes the element is connected to (the nodes of elements with more than two nodes are given on succeeding lines).

PARAMETERS:

select-elements Select elements; see Section 5.1.

PRINT HINGE

...	HINGE	select-elements
-----	-------	-----------------

PURPOSE:

The command prints a table of the hinges of the elements. The table has the following appearance:

SUPER ELEMENT TYPE:			1	LEVEL:		1			
EXT.			H I N G E		V A L U E S				
EL.	ND	TX	TY	TZ	RX	RY	RZ	CS	CODE
122	1	1.000	1.000	0.000	0.000	1.000	0.000	G	ESTIF
	2	6.000	5.000	INFINITY	INFINITY	2.000	1.000	L	ESTIF
324	1	0.100	0.200	0.300	0.400	0.500	0.600	L	DFIX
	2	1.000	1.000	0.000	0.000	1.000	0.000	L	DFIX
422	1	0.100	0.200	30.000	40.000	400.000	7777.000	G	ESTIF
	2	1.000	1.000	0.000	0.000	1.000	0.000	L	DFIX

columns of the table give, from left to right:

- user defined (external) element number,
- the first and second node of the element,
- hinge values (coefficients) for the six d.o.f.s,
- coordinate system of hinge values:
 - G = global
 - L = local
- hinge code (see the PROPERTY HINGE command):
 - DFIX = degree of fixation (the hinge values are coefficients between 0 and 1)
 - ESTIF = inter-element elastic spring stiffness (the hinge values are stiffnesses)

PARAMETERS:

select-elements Select elements; see Section 5.1.

PRINT LOAD

...	LOAD	ALL-LOADCASES	...
		OVERVIEW	
		SUM	
		load-case	

PURPOSE:

The command prints tables over the loads.

The ALL-LOADCASES option prints all load cases. The loads are primarily sorted by load type, secondarily by load case, and thirdly by node/element number.

PRINT LOAD OVERVIEW

...	OVERVIEW	GENERAL	...	load-case
		INERTIA		ALL-LOADCASES
		NODE		END
		ELEMENT		
		END		

PURPOSE:

The command prints a selected load overview table. The tables have the following appearances:

Example of general load overview

```

SUPER ELEMENT TYPE:      1  LEVEL:      1
      N U M B E R   O F   L O A D S
LOADCASE   INERTIA      NODAL      ELEMENTAL
-----
          2              1          1          6
          5              10         7

```

Example of inertia load overview

```

SUPER ELEMENT TYPE:      1  LEVEL:      1
      N U M B E R   O F   I N E R T I A   L O A D S
LOADCASE   GRAVITY      ROT. ACC.
-----
          2              1
          5

```

Example of node load overview

```

SUPER ELEMENT TYPE:      1  LEVEL:      1
      N U M B E R   O F   N O D A L   L O A D S
      FORCES   PRESCR.   PRESCR.   ROT.   TEMP.   TEMP.
LOADCASE  MOMENTS  DISPL.   ACC.   ACC.   TEMP.   1 DERIV.  2 DERIV.
-----
          2              1
          5              6          2          2

```

Example of element load overview

```

SUPER ELEMENT TYPE:      1  LEVEL:      1
      N U M B E R   O F   E L E M E N T A L   L O A D S
      DISTRIB.  POINT ON  UNIFORM  UNIFORM  TEMPE-
LOADCASE  BEAM   BEAM   LINE   SURFACE  VOLUME  RATURE
-----
          2              6
          5              3          4

```

PRINT LOAD SUM

...	SUM	YES	...	load-case
		NO		ALL-LOADCASES
				END

PURPOSE:

The command prints load sum tables. Either the sum of positive and negative values separately and total sum (answer YES), or the total sum alone (answer NO), are printed. The load sum calculation takes into account the load types: gravity, nodal forces, element point forces and element distributed forces. The table has the following appearance:

		X	Y	Z
		-----	-----	-----
L O A D C A S E	:	5		
R E A L				
SUM OF GIVEN FORCES				
	POSITIVE :	135.5371	208.1616	266.5698
	NEGATIVE :	-19.8949		
	TOTAL :	115.6423	208.1616	266.5698
SUM OF GIVEN MOMENTS				
	POSITIVE :	90.9959	107.7828	121.1479
	NEGATIVE :	-2.2817		
	TOTAL :	88.7142	107.7828	121.1479
SUM OF MOMENTS FROM GIVEN FORCES ABOUT GLOBAL AXES				
	POSITIVE :		10490.6943	1333.7736
	NEGATIVE :	-15041.2344	-1449.0376	-1187.7146
	TOTAL :	-15041.2344	9041.6572	146.0590
SUM OF MOMENTS FROM GIVEN FORCES AND GIVEN MOMENTS				
	POSITIVE :	90.9959	10598.4775	1454.9214
	NEGATIVE :	-15043.5156	-1449.0376	-1187.7146
	TOTAL :	-14952.5205	9149.4404	267.2068

The table is repeated for any imaginary contribution to the load case(s).

PRINT LOAD load-case

...	load-case	ALL-TYPES-NODES-ELEMENTS	
		GRAVITY	
		ROTATION-OF-STRUCTURE	
		NODE-FORCE	select-nodes
		NODE-PRESCRIBED	select-nodes
		ELEMENT-FORCE	select-elements
		ELEMENT-TEMPERATURE	select-elements
		END	

PURPOSE:

The command prints a selected set of basic loads for one load case. The loads are primarily sorted by load type, secondarily by load case, and thirdly by node/element number. The tables have the following appearances:

Example of gravity load

```

L O A D C A S E           :                               2
CONTRIBUTION              : FROM COMPLETE ELEMENT
GRAVITY      TX TY TZ    :                               -9.8100

```

Example of rotation-of-structure load

```

L O A D C A S E           :                               7
ROTATION-OF-STRUCTURE     :
  AXIS POS. P1  X1 Y1 Z1 :          -20.0000          20.0000
  AXIS POS. P2  X2 Y2 Z2 :          -10.0000          10.0000      100.0000
  AXIS DIR.    DX DY DZ :           0.0990          -0.0990           0.9901
  ANGULAR VELOCITY (RAD) :                               10.0000
  ANGULAR ACCEL.   (RAD) :

```

Example of node force loads

```

L O A D C A S E           :                               5
NODE      INDEX  LOADCASE :              701              1              5
FORCE      TX TY TZ :          33.0000          44.0000          55.0000  REAL
MOMENT      RX RY RZ :          77.0000          88.0000          99.0000  REAL
NODE      INDEX  LOADCASE :              701              2              5
FORCE      TX TY TZ :           1.0000           2.0000           3.0000  REAL
MOMENT      RX RY RZ :           4.0000           5.0000           6.0000  REAL
              TX TY TZ :           7.0000           8.0000           9.0000  IMAG.
              RX RY RZ :           1.0000           2.0000          2.0000  IMAG.

```

Example of node prescribed displacement/acceleration loads

```

L O A D C A S E           :                               5

```

NODE	INDEX	LOADCASE :	304	1	5	
PRE. DISPL.		TX TY TZ :	1.0000	2.0000		REAL
		RX RY RZ :	3.0000	4.0000		REAL
NODE	INDEX	LOADCASE :	308	1	5	
PRE. ACC.		TX TY TZ :	1.0000	2.0000		REAL
		RX RY RZ :	4.0000	5.0000		REAL

Example of element distributed loads

L O A D C A S E	:				5	
ELEMENT INDEX	LOADCASE :	222	1		5	
DISTANCE FROM	E1 E2 :	1.0000				FROM PROJ. OF NODES
FORCE END 1	TX TY TZ :	-2.0616	0.7668		3.0268	REAL
FORCE END 2	TX TY TZ :	-0.0904	2.7380		2.5485	REAL
ELEMENT INDEX	LOADCASE :	332	1		5	
DISTANCE FROM	E1 E2 :	2.0000				FROM PROJ. OF NODES
FORCE END 1	TX TY TZ :	4.0000	5.0000		6.0000	AMPL.
	TX TY TZ :	2.0000	2.0000		4.0000	PH-DEG
FORCE END 2	TX TY TZ :	4.0000	5.0000		6.0000	AMPL.
	TX TY TZ :	4.0000	4.0000		2.0000	PH-DEG

Example of element point loads

L O A D C A S E	:				5	
ELEMENT INDEX	LOADCASE :	704	1		5	
DISTANCE FROM	E1 :	1.0000				FROM PROJ. OF NODES
POINT FORCE	TX TY TZ :	1.0000	2.0000		3.0000	REAL
	TX TY TZ :	3.0000	2.0000		1.0000	IMAG.
ELEMENT INDEX	LOADCASE :	704	2		5	
DISTANCE FROM	E1 :	3.0000				FROM PROJ. OF NODES
POINT FORCE	TX TY TZ :	3.0000	2.0000		1.0000	AMPL.
	TX TY TZ :	2.0000	4.0000		6.0000	PH-DEG
ELEMENT INDEX	LOADCASE :	714	1		5	
DISTANCE FROM	E1 :	4.0000				FROM PROJ. OF NODES
POINT FORCE	TX TY TZ :	5.0000	6.0000		7.0000	REAL

Example of element temperature loads

L O A D C A S E	:				6	
ELEMENT INDEX	LOADCASE :	231	1		6	
TEMP. DIFF.	NODE T1 :	204	1.0000			
TEMP. DIFF.	NODE T1 :	304	9.0000			
ELEMENT INDEX	LOADCASE :	702	1		6	
TEMP. DIFF.	NODE T1 :	701	15.0000			
TEMP. DIFF.	NODE T1 :	703	15.0000			

PARAMETERS:

load-case	Local load case number.
select-elements	Select elements; see Section 5.1.
select-nodes	Select nodes; see Section 5.1.

PRINT LOCAL-COORDINATE

...	LOCAL-COORDINATE	select-elements
-----	------------------	-----------------

PURPOSE:

The command prints a table of the local coordinate systems of the elements. The table has the following appearance:

SUPER ELEMENT TYPE:				1	LEVEL:		1					
EXT.	LOCAL-X				LOCAL-Y				LOCAL-Z			
EL.	ND	GX	GY	GZ	GX	GY	GZ	GX	GY	GZ	REM.	
101	1	0.099	-0.099	0.990	0.707	0.707		-0.700	0.700	0.140	CALC.	
102	1	1.000		0.000		1.000		0.000		1.000	CALC.	
234											NONE	
235											NONE	
734	1	0.844	-0.053	-0.534	0.537	0.084	0.840		-0.995	0.100	SPEC.	
735	1	0.844	0.053	0.534	-0.537	0.084	0.840		-0.995	0.100	SPEC.	
1010	1	1.000				1.000				1.000	GLOB.	
1020	1	1.000				1.000				1.000	GLOB.	
9005	1	-0.099	-0.099	0.990	0.707	-0.707		0.700	0.700	0.140	T9005	
9008	1	0.707		-0.707		1.000		0.707		0.707	T9008	

columns of the table give, from left to right:

- user defined (external) element number,
- ND is a redundant value and should be ignored,
- direction of local x-axis in global coordinate system,
- direction of local y-axis in global coordinate system,
- direction of local z-axis in global coordinate system,
- REM. = remark, the remarks have the following interpretation:
 - NONE = element has no local coordinate system
 - SPEC. = local coordinate system specified by the user (PROPERTY LOCAL-COORDINATE)
 - CALC. = local coordinate system calculated by the program (default local coordinate system)
 - GLOB. = local coordinate system is the same as the global coordinate system
 - Tnnnn = local coordinate system is the same as the transformed coordinate system corresponding to transformation number nnnn
 - ERR-L = Error! Element length is zero.
 - ERR- = Error! Local coordinate system is not orthonormal. Element axis has been changed after local coordinate system was defined. E.g. if an eccentricity involving a rotation is introduced, or the position of a node is changed after specification of the local coordinate system

PARAMETERS:

select-elements Select elements; see Section 5.1.

PRINT MASS-OF-ELEMENTS

...	MASS-OF-ELEMENTS	select-elements
-----	------------------	-----------------

PURPOSE:

The command prints mass (weight) of selected beam elements. The table has the following appearance:

CENTROID OF STRUCTURAL ELEMENTS				
	X	Y	Z	
				0.0000
				0.0000
				62.4814
MASS MOMENT OF INERTIA OF STR.				
EL. ABOUT CENTROID	X	Y	Z	
				220349.2031
				313812.5938
				448066.3750
MASS MOMENT OF INERTIA OF STR.				
EL. ABOUT ORIGIN	X	Y	Z	
				1917869.1250
				2011332.5000
				448066.3750
MASS OF STRUCTURAL ELEMENTS				
				434.8236
NO. OF ELEMENTS CONTRIBUTING				
				64

PARAMETERS:

select-elements Select elements; see Section 5.1.

NOTES:

Print commands are as default not logged. Activate logging by the command SET JOURNALLING PRINT ON.

PRINT MATERIAL

...	MATERIAL	matno
		ALL
		OVERVIEW
		END

PURPOSE:

The command prints material data. Data for a single or all material numbers may be printed, or a simple overview of the materials may be printed. The tables have the following appearances:

```

MATERIAL NUMBER :           3
MATERIAL TYPE   :   Spring to ground matrix
Matrix type                                           FLEXIBILITY
No. of degrees of freedom                               6
0.1000E+01
0.0000E+00   0.2000E+01
0.0000E+00   0.0000E+00   0.3000E+01
0.0000E+00   0.0000E+00   0.0000E+00   0.4000E+01
0.0000E+00   0.0000E+00   0.0000E+00   0.0000E+00   0.5000E+01
0.0000E+00   0.0000E+00   0.0000E+00   0.0000E+00   0.0000E+00   0.6000E+01

MATERIAL NUMBER :           4
MATERIAL TYPE   :   Linear isotropic elastic, structural analysis
Young's modulus                                0.2100E+12
Poisson's ratio                                0.3000E+00
Density                                           0.7850E+04
Thermal expansion coefficient                    0.1200E-04

MATERIAL NUMBER :           6
MATERIAL TYPE   :   Axial spring
Axial spring constant                            0.3450E+03
    
```

PARAMETERS:

matno Material reference number.

PRINT NODE

...	NODE	BOUNDARY-CONDITION	...
		COORDINATE	
		INITIAL-CONDITION	
		LINEAR-DEPENDENCY	
		MASS-ON-NODE	

PURPOSE:

The command prints different tables of data related to the nodes.

PRINT NODE BOUNDARY-CONDITIONS

...	BOUNDARY-CONDITIONS	select-nodes
-----	---------------------	--------------

PURPOSE:

The command prints a table of the boundary conditions of the nodes. The table has the following appearance:

SUPER ELEMENT TYPE:		1	LEVEL:		1				
EXT.	INT.	TRANSF	BOUNDARY CONDITIONS						
NO.	NO.	NO.	TX	TY	TZ	RX	RY	RZ	
104	22		SUPERL		SUPERL	SUPERL			
304	26		PRESC.	PRESC.		PRESC.	PRESC.		
405	12	888	FIXED			FIXED			
701	3		LINEAR	LINEAR	LINEAR	LINEAR	LINEAR	LINEAR	
703	19		SUPER	SUPER	SUPER	X	X	X	
705	4		LINEAR				LINEAR		
708	23		SUPER		SUPER		SUPER		

The columns of the table give, from left to right:

- user defined (external) node number,
- internal node number (initially, the first node created is number 1; the last is N where N is the number of nodes, optimising the node numbering will change this), this number is normally of no interest to the user,
- transformation reference number, the boundary conditions of the node relate to the transformed coordinate system instead of the global coordinate system,
- boundary condition codes for the six d.o.f.s

It is possible to switch between text and digits for boundary condition codes by the SET PRINT TABLE NODE-BOUNDARY-TABLE command. The possible boundary condition codes are:

<u>digit</u>	<u>text</u>	<u>boundary condition of d.o.f.</u>
-1	X	does not exist (the node has reduced d.o.f.s)
0	(blank)	free
1	FIXED	fixed at zero displacement
2	PRESC.	prescribed displacement
3	LINEAR	linearly dependent of some other d.o.f.(s)
4	SUPER	super d.o.f.
100	SUPERL	super d.o.f. due to linear dependency

PARAMETERS:

select-nodes Select nodes; see Section 5.1.

PRINT NODE COORDINATES

...	COORDINATES	select-nodes
-----	-------------	--------------

PURPOSE:

The command prints a table of the coordinates of the nodes. The table also indicates whether boundary conditions have been defined for the nodes and gives the number of d.o.f.s of the nodes. The table has the following appearance:

SUPER ELEMENT TYPE:		1 LEVEL:		1	
EXT.	INT.	C O O R D I N A T E S			BOU
NO.	NO.	X	Y	Z	CON ND
101	1	-20.000000	20.000000	0.000000	X 6
104	22	-20.000000	-20.000000	0.000000	X 6
105	2	20.000000	20.000000	0.000000	X 6
108	21	20.000000	-20.000000	0.000000	6
201	5	-19.459459	19.459459	5.405406	6
202	41	-19.459461	0.000006	5.405409	6
705	4	10.000000	10.000000	100.000000	X 6
706	39	0.000000	-10.000002	100.000000	3
707	51	10.000000	0.000000	100.000000	X 3
708	23	10.000000	-10.000000	100.000000	X 6

The columns of the table give, from left to right:

- user defined (external) node number,
- internal node number (initially, the first node created is number 1; the last is N where N is the number of nodes, optimising the node numbering will change this), this number is normally of no interest to the user,
- the coordinates given in the global coordinate system,
- BOU CON = boundary condition, an 'X' indicates that some boundary condition has been defined for some of the d.o.f.s of the node, 'blank' indicates that all d.o.f.s of the node are FREE,
- ND = number of d.o.f.s of the node.

The format of the print of the coordinates may be changed by the SET PRINT FORMAT command.

PARAMETERS:

select-nodes Select nodes; see Section 5.1.

PRINT NODE INITIAL-CONDITION

...	INITIAL-CONDITION	select-nodes
-----	-------------------	--------------

PURPOSE:

The command prints a table of the initial conditions defined for the nodes. The table has the following appearance:

SUPER ELEMENT TYPE:		1		LEVEL:		1	
EXT.	INT.	I N I T I A L		C O N D T I O N			
NO.	NO.	RX	RY	RZ	TYPE		

101	1	5.000000	6.000000	7.000000	VELO.		
		8.000000	9.000000	10.000000			
105	2	1.000000	0.000000	8.000000	DISP.		
		5.000000	5.000000	0.000000			
105	2	1.000000	2.000000	3.000000	VELO.		
		4.000000	5.000000	6.000000			

columns of the table give, from left to right:

- user defined (external) node number,
- internal node number (initially, the first node created is number 1; the last is N where N is the number of nodes, optimising the node numbering will change this), this number is normally of no interest to the user,
- the initial conditions, the first line for each node gives the initial condition of the translational d.o.f.s while the second line gives the initial condition for the rotational d.o.f.s.
- type of initial condition, displacement or velocity.

The format of the print of the initial conditions may be changed by the SET PRINT FORMAT command.

PARAMETERS:

select-nodes Select nodes; see Section 5.1.

PRINT NODE LINEAR-DEPENDENCY

...	LINEAR-DEPENDENCY	select-nodes
-----	-------------------	--------------

PURPOSE:

The command prints a table of the linear dependency factors of nodes (for which such are defined). The table has the following appearance:

SUPER ELEMENT TYPE:		1	LEVEL:		1			
LIN.DEP.	INDEP.				DEPENDENT	D.O.F.		
NODE	NODE		TX	TY	TZ	RX	RY	RZ
-----	-----		-----	-----	-----	-----	-----	-----
701	704	TX	0.300					
		TY		0.300				
		TZ			0.300			
		RX				0.300		
		RY					0.300	
		RZ						0.300
705	104	TX					0.700	
		TY						
		TZ	0.440				1.500	
		RX	0.550					
		RY						
		RZ						
	105	TX	0.600					
		TY						
		TZ						
		RX					0.800	
		RY						
		RZ						

The columns of the table give, from left to right:

- user defined (external) node number of the linearly dependent node,
- user defined (external) node number of the independent node,
- the linear dependency factors organised in a matrix where each column represent the dependency of a linearly dependent d.o.f. of the independent d.o.f.s.

In the example above node 701 is linearly dependent of node 704 only while node 705 is linearly dependent of the two nodes 104 and 105.

PARAMETERS:

select-nodes Select nodes; see Section 5.1.

PRINT NODE MASS-ON-NODE

...	MASS-ON-NODE	select-nodes
-----	--------------	--------------

PURPOSE:

The command prints a table of the nodal masses defined. The table has the following appearance:

SUPER ELEMENT TYPE:		1	LEVEL:				1
EXT.	INT.		M	A	S	S	
NO.	NO.	X	Y				Z

401	7	1.000000	2.000000				3.000000
		4.000000	5.000000				6.000000
402	43	1.000000	2.000000				3.000000
		4.000000	5.000000				6.000000
408	32	6.000000	5.000000				4.000000
		3.000000	2.000000				1.000000

columns of the table give, from left to right:

- user defined (external) node number,
- internal node number (initially, the first node created is number 1; the last is N where N is the number of nodes, optimising the node numbering will change this), this number is normally of no interest to the user,
- the nodal masses, the first line for each node gives the translational nodal masses while the second line gives the rotational nodal masses.

The format of the print of the masses can be changed by the SET PRINT FORMAT command.

PARAMETERS:

select-nodes Select nodes; see Section 5.1.

PRINT SECTION

...	SECTION	sctno	YES
			NO
		ALL	YES
			NO
		OVERVIEW	
		END	

PURPOSE:

The command prints cross sectional data. Data for a single or all section numbers may be printed, or a simple overview of the sections. Only parameters defining the section may be printed (answer NO) or also data computed for the section (area and moments of inertia) (answer YES). The tables have the following appearances:

The overview:

SECTION NO	SECTION TYPE	DIAM. / HEIGHT / TH.
2	CHANNEL	4.000000
11	PIPE	12.000000
22	I	7.000000
99	GENERAL	

Data for a single section including data computed for the section:

SECTION NUMBER :	2	
SECTION TYPE :	CHANNEL	
HZI	HEIGHT AT END	4.000000
BY	FLANGE WIDTH	3.000000
TZ	FLANGE THICKNESS	1.000000
TY	WEB THICKNESS	2.000000
SFY	SHEAR FACTOR Y DIRECTION	1.000000
SFZ	SHEAR FACTOR Z DIRECTION	1.000000
K	WEB LOCATION IN LOCAL Y-DIRECTION	POSITIVE
AREA	CROSS SECTION AREA	10.000000
IX	TORSIONAL MOMENT OF INERTIA ABOUT SHEAR CENTRE	8.213333
IY	MOMENT OF INERTIA ABOUT Y AXIS	15.333333
IZ	MOMENT OF INERTIA ABOUT Z AXIS	6.433334
IYZ	PRODUCT OF INERTIA ABOUT Y AND Z AXES	0.000000
WXMIN	MIN. TORSIONAL SECTION MODULUS ABOUT SHEAR CENTRE	4.106667
WYMIN	MIN. SECTION MODULUS ABOUT Y AXIS	7.666667
WZMIN	MIN. SECTION MODULUS ABOUT Z AXIS	3.784314
SHARY	SHEAR AREA IN THE DIRECTION OF Y AXIS	4.452134
SHARZ	SHEAR AREA IN THE DIRECTION OF Z AXIS	5.575758
SHCENY	SHEAR CENTRE LOCATION FROM CENTROID Y COMPONENT	0.966667
SHCENZ	SHEAR CENTRE LOCATION FROM CENTROID Z COMPONENT	0.000000

SY	STATIC AREA MOMENT ABOUT Y AXIS	5.500000
SZ	STATIC AREA MOMENT ABOUT Z AXIS	2.890000
CY	CENTROID LOC. FROM BOTTOM RIGHT CORNER Y COMPONENT	1.700000
CZ	CENTROID LOC. FROM BOTTOM RIGHT CORNER Z COMPONENT	2.000000

PRINT SOIL

...	SOIL	PY-TZ-QZ-CODE
		SKIN-FRICTION
		TIP-RESISTANCE
		TYPE

PURPOSE:

The command prints an overview of the defined data sets for the selected data types:

Example of print of PY-TZ-QZ-CODE data:

```

                                SOIL DATA
                                PY TZ QZ -CODE
                                =====
SUPER ELEMENT TYPE:           1  LEVEL:           1

Z LEVEL      PY-CODE      TZ-CODE      QZ-CODE
-----
      -1.50          287          293          293

```

Example of print of SKIN-FRICTION data:

```

                                SOIL DATA
                                SKIN FRICTION
                                =====
SUPER ELEMENT TYPE:           1  LEVEL:           1

Z LEVEL      PEAK SKIN FRICTION IN  INIT. VALUE DSPTZ/  TIP PEAK      POISS. DSPQZ/
              COMPRESSION  TENSION  SHEAR MOD.  DIAM      STRESS      RATIO  DIAM
-----
      -1.50    0.5000E+01  0.3000E+01 -0.1000E+01   0.01    0.0000E+00   0.50   0.05
      -3.50    0.1500E+02  0.1100E+02 -0.1000E+01   0.01    0.0000E+00   0.50   0.05
      -5.50    0.4500E+02  0.4500E+02 -0.1000E+01   0.01    0.0000E+00   0.50   0.05
     -14.50    0.7500E+02  0.7500E+02 -0.1000E+01   0.01    0.0000E+00   0.50   0.05
     -27.50    0.1100E+03  0.9500E+02 -0.1000E+01   0.01    0.0000E+00   0.50   0.05

```

Example of print of TIP-RESISTANCE data:

```

                                SOIL DATA
                                TIP RESISTANCE
                                =====
SUPER ELEMENT TYPE:           1  LEVEL:           1

              PEAK SKIN FRICTION IN  INIT. VALUE DSPTZ/  TIP PEAK      POISS. DSPQZ/

```

Z LEVEL	COMPRESSION	TENSION	SHEAR MOD.	DIAM	STRESS	RATIO	DIAM
-36.50	0.1200E+03	0.1200E+03	-0.1000E+01	0.01	0.1335E+05	0.50	0.05
-101.50	0.1200E+03	0.1200E+03	-0.1000E+01	0.01	0.1400E+05	0.50	0.05

Example of print of TYPE data:

SOIL DATA
SOIL TYPE
=====

SUPER ELEMENT TYPE: 1 LEVEL: 1

SOI	TOTAL	FRICT	UNDRAINED	SHEAR AT		OVER		GAP	TRES/	TZZ	
TYP	UNIT	WEIGHT	ANGLE	Z = 0.0	Z = 100.0	EPSC	CONS.	API-J	VAL	TMAX	RES
---	-----	-----	-----	-----	-----	-----	-----	-----	---	-----	-----
1	0.1950E+02	38.00	0.0000E+00	0.0000E+00	0.00	1.00	0.00	0	1.00	0.005	
2	0.1950E+02	32.00	0.0000E+00	0.0000E+00	0.00	1.00	0.00	0	1.00	0.005	
3	0.1900E+02	0.00	0.1000E+03	0.1000E+03	0.01	1.00	0.50	0	1.00	0.005	
4	0.1900E+02	0.00	0.1300E+03	0.1300E+03	0.01	1.00	0.50	0	1.00	0.005	
5	0.2000E+02	37.00	0.0000E+00	0.0000E+00	0.00	1.00	0.00	0	1.00	0.005	

NOTES:

See also the Gensod User Manual for specific soil related explanation.

PRINT STATUS

...	STATUS	YES
		NO

PURPOSE:

The command prints a table summarising key data for the model. Optionally (answer YES) additional data is printed: coordinates of the centroid (centre of gravity) not including and including nodal masses, mass moment of inertia about the centroid, mass moment of inertia about the origin, mass sum not including nodal masses, and finally mass sum for nodal masses. This calculation only takes into account beam, truss and non-structural beam elements for which materials and cross sections are given; only the flexible part of the elements contribute in the calculation. The formulae for calculating the centroid and the mass moment of inertia assume that the mass of the elements is distributed evenly along the element axis, i.e. the shape of the cross sections are not taken into account. The table has the following appearance:

MODEL FILE	PREFRAME1.MOD		
LOG FILE	PREFRAME1.JNL		
SUPER ELEMENT TYPE			1
COORDINATE-TOLERANCE		0.1000	
ANGLE-TOLERANCE		0.0010	
UNIT-VECTOR-TOLERANCE		0.0010	
ELEMENTS	TOTAL		131
	BEAM (BEAS)		106
	TRUSS (TESS)		12
	SPRING TO GROUND (GSPR)		2
	OTHER		11
NODES	TOTAL		52
	FIXED		1
	PRESCRIBED DISPL.		3
	LINEAR DEPENDENT		3
	SUPER		8
SECTIONS			7
MATERIALS			7
LOAD CASES	TOTAL		4
	INERTIA		2
	NODE		2
	ELEMENT		3
CENTROID OF STRUCTURAL ELEMENTS			
	X Y Z	-0.4593	-0.0415 56.3558
MASS MOMENT OF INERTIA OF STR.			
EL. ABOUT CENTROID	X Y Z	0.9350627E+12	0.9377428E+12 0.3145493E+12
MASS MOMENT OF INERTIA OF STR.			
EL. ABOUT ORIGIN	X Y Z	0.3884492E+13	0.3887367E+13 0.3147469E+12
MASS OF STRUCTURAL ELEMENTS		928669120.0000	
NO. OF ELEMENTS CONTRIBUTING		118	
SUM OF NODE MASSES	TX TY TZ	16.0000	18.0000 20.0000
	RX RY RZ	22.0000	24.0000 26.0000
NO. OF NODES CONTRIBUTING		6	
CENTROID OF STRUCTURAL ELEMENTS			

AND	NODE	MASS	TX	X	Y	Z	-0.4593	-0.0415	56.3558
AND	NODE	MASS	TY	X	Y	Z	-0.4593	-0.0415	56.3558
AND	NODE	MASS	TZ	X	Y	Z	-0.4593	-0.0415	56.3558

PRINT STRUCTURE-CONCEPT

...	STRUCTURE-CONCEPT	ALL	...	ALL
		MEMBER		PARENT-ONLY
		PILE		
		JOINT	...	ALL
				MEMBER
				PILE
		MEMBER-INCIDENCES		

PURPOSE:

The command prints various tables of the structure concepts. The table has the following appearance:

Example of print member data

SUPER ELEMENT TYPE: 1 LEVEL: 1

----- STRUCTURE CONCEPT -----					# PART	CONCEPT NO.		EXT. NUMBER	
NO.	NAME	TYPE	ROLE	PARENT	CONC.	START	END	ELEMENT	NODE
	1 301422	MEMBER	UNDEF.	NONE	2	4	5		
	2	SEGM.	MIDSEC		1			301422	
	3	SEGM.	STUB		1			302433	
	4	JOINT	UNDEF.	NONE					301421
	5	JOINT	UNDEF.	NONE					10221
	6 301122	MEMBER	UNDEF.	NONE	2	9	5		
	7	SEGM.	MIDSEC		6			301122	
	8	SEGM.	STUB		6			302434	
	9	JOINT	UNDEF.	NONE					301121
	10 202121	MEMBER	UNDEF.	NONE	2	13	5		
	11	SEGM.	MIDSEC		10			202121	
	12	SEGM.	STUB		10			302435	
	13	JOINT	UNDEF.	NONE					10211

Example of print pile data

SUPER ELEMENT TYPE: 1 LEVEL: 1

----- STRUCTURE CONCEPT -----					# PART	CONCEPT NO.		EXT. NUMBER	
NO.	NAME	TYPE	ROLE	PARENT	CONC.	START	END	ELEMENT	NODE

43	P71	PILE	MAIN	NONE	5	49	50		
44		PILELE	MIDSEC		43			120	
45		PILELE	MIDSEC		43			121	
46		PILELE	MIDSEC		43			122	
47		PILELE	MIDSEC		43			123	
48		PILELE	MIDSEC		43			124	
49		PILJOI	UNDEF.	NONE					112

50
 PILJOI UNDEF. NONE
 117

PARAMETERS:

ALL	Print all available data.
MEMBER	Print for member concepts only.
PILE	Print for pile concepts only.
JOINT	Print for joint concepts only.
PARENT-ONLY	Print parent conceptual information only.
MEMBER-INCIDENCES	Print member concepts with corresponding start and end nodes.

PRINT TRANSFORMATION

...	TRANSFORMATION	trano
		ALL
		END

PURPOSE:

The command prints the transformation matrices of the user defined transformations (the TRANSFORMATION command). A single or all transformation matrices may be printed. The table has the following appearance:

```
Transformation number:      888
  T:      0.577      -0.707      -0.408      0.000
          0.577       0.707      -0.408      0.000
          0.577       0.000       0.816      0.000
```

The fourth column has no meaning in this context and should be disregarded.

PARAMETERS:

trano Transformation reference number.

PROPERTY

PROPERTY	CONNECT	...
	ECCENTRICITY	
	GAP	
	HINGE	
	LOCAL-COORDINATE	
	MATERIAL	
	SECTION	
	SOIL	

PURPOSE:

The command defines properties. There are six types of properties:

- eccentricities or offsets, defined explicitly by the PROPERTY ECCENTRICITY command, or implicitly (computed by the program) by the PROPERTY GAP command,
- hinged connection of elements to nodes,
- local coordinate systems of elements,
- geometrical cross sectional data,
- material data,
- soil data.

Cross sections and material data are first defined and given reference numbers, and subsequently assigned to the elements by the PROPERTY CONNECT command.

The other properties are defined directly for the relevant elements.

PROPERTY CONNECT

...	CONNECT	SECTION	sctno	select-elements
		MATERIAL	matno	

PURPOSE:

The command connects or assigns cross sections and materials to elements.

The PROPERTY CONNECT command may be repeated to override a previous assignment for an element.

PARAMETERS:

SECTION	A cross section is to be connected.
sctno	Number of the cross section to be connected.
MATERIAL	A material is to be connected.
matno	Number of the material to be connected.
select-elements	Select elements; see Section 5.1.

PROPERTY ECCENTRICITY

...	ECCENTRICITY	select-elements	{	GLOBAL		ex	ey	ez	}*2
				LOCAL	elno				
				TRANSFORMATION	trano				

PURPOSE:

The command defines eccentricities (offsets) for selected elements. The eccentricities are defined for the two element ends.

Eccentricities can be changed by the CHANGE ECCENTRICITY command (or by this command), and deleted by the DELETE ECCENTRICITY command.

PARAMETERS:

select-elements	Select elements; see Section 5.1.
GLOBAL	The offset values refer to the global coordinate system.
LOCAL	The offset values refer to a local coordinate system.
elno	The local coordinate system to be applied belongs to element elno.
TRANSFORMED	The transformed coordinate system to be applied belongs to transformation trano.
trano ex ey ez	The offset values at the end, given as a vector from the node to the beam end.

NOTES:

Introducing eccentricities for an element for which a local coordinate system has previously been defined (using the PROPERTY LOCAL-COORDINATE command) will lead to error if the eccentricity involves changing the direction of the local x-axis. This because the local y- and z-axes have been fixed by the PROPERTY LOCAL-COORDINATE command and changing the x-axis will then give a non-cartesian coordinate system. Printing the local coordinate system for the element will reveal this by a remark (ERROR) in the right column of the table printed.

See the SET ELEMENT-LOAD-DISTANCE-MODE command for an explanation of the effect of distributed load on an element with eccentricities.

PROPERTY GAP

...	GAP	node	chord-el	aligned-el	legal-gap	...
...	FIXED-BRACE			element-no		...
	IGNORE-BRACE					
	SYMMETRIC-ELEMENT			element-no	element-no	
...	FIXED-BRACE		element-no			
	GAP		element-no	element-no	legal-gap	
	END					

PURPOSE:

The command introduces eccentricities for braces in a tubular joint.

The use of the command is explained in detail in Section 3.6.3.

PARAMETERS:

node	Node number for which the eccentricities shall be introduced
chord-el	The chord element
aligned-el	The aligned element
legal-gap	The minimum allowable gap
FIXED-BRACE	Select a brace for which the eccentricity shall not be changed. Other braces will be moved with respect to this.
IGNORE-BRACE	Select a brace which shall not be moved and which will not influence other braces. The brace(s) may only be selected before the first fixed brace is specified.
SYMMETRIC-ELEMENTS	Select two braces which shall be moved equally away from each other if the gap is too small.
GAP	Specify a special gap between a pair of elements.
END	The specification is complete. The calculation will start and some execution time should be expected.

NOTES:

See the warning about introducing eccentricities for an element for which a local coordinate system has previously been defined given for the command PROPERTY ECCENTRICITY.

PROPERTY HINGE

...	HINGE	select-elements	{	GLOBAL	...
				LOCAL	

...	FIXATION-(CONNECTIVITY)		alpha _i *6		}*2
	INTERELEMENT-ELASTIC-SPRING-STIFFNESS		{ c _i	}*6	
			{ INFINITY		

PURPOSE:

The command introduces hinges involving d.o.f.s of element ends to be released from the d.o.f.s of the nodes. Hinge data is given for the two beam nodes individually.

Also see Section 3.3.2.

A hinge can be changed by the CHANGE HINGE command, and deleted by the DELETE HINGE command.

For each d.o.f. of the two element ends the user gives the coefficient of fixation (connectivity) ($0 \leq \alpha_i \leq 1$) or alternatively, the inter-element elastic spring stiffness ($c_i \geq 0$). The hinge coefficients can either be specified in the GLOBAL coordinate system or in the LOCAL element coordinate system.

The relationship between c_i and α_i is

$$\alpha_i = c_i / (k_{ii} + c_i)$$

where k_{ii} is the diagonal term of the element stiffness matrix corresponding to d.o.f. number i of the current node.

$\alpha_i = 0$ is fully released

$\alpha_i = 1$ is fully connected (no hinge)

PARAMETERS:

select-elements	Select elements; see Section 5.1.
GLOBAL	The hinge coefficients refer to the global coordinate system.
LOCAL	The hinge coefficients refer to the element's local coordinate system.
FIXATION-(CONNECTIVITY)	Coefficients of fixation (connectivity) are to be given.

α_i	The coefficient of fixation, α_i , of the i'th d.o.f.
INTERELEMENT-ELASTIC-SPRING-STIFFNESS	Inter-element elastic spring stiffnesses are to be given.
c_i	Inter-element elastic spring stiffness, c_i , of the i'th d.o.f.
INFINITY	Inter-element elastic spring stiffness of the i'th d.o.f. is given an infinitely high value.

PROPERTY LOCAL-COORDINATE

...	LOCAL-COORDINATE	YX-PLANE	...
		ZX-PLANE	

...	X-GLOBAL-INFINITY			select-elements
	X-GLOBAL-INFINITY			
	Y-GLOBAL-INFINITY			
	Y-GLOBAL-INFINITY			
	Z-GLOBAL-INFINITY			
	Z-GLOBAL-INFINITY			
	GUIDING-POINT	gx	gy	gz
	PLANE	node1	node2	node3

PURPOSE:

The command defines local coordinate systems for elements. The command may be repeated to override previous definitions. Local coordinate systems cannot be deleted or changed.

All beam and non-structural beam elements will have local coordinate systems whether they are implicitly defined (calculated by the program) or explicitly defined (by the PROPERTY LOCAL-COORDINATE command). The local x-axis is by definition the neutral axis of the cross section and pointing from beam end 1 towards beam end 2. Beam ends 1 and 2 are implicitly defined when creating the element: end 1 is the first node given when creating the element. The local y-z-plane is normal to the local x-axis, and defining a local coordinate system involves determining the orientation of the local y- and z-axes.

The orientation of the local y- and z-axis is defined by either determining the orientation of the y-x-plane or the z-x-plane. The orientation of either of these planes is defined by either of the following two methods:

- A guiding point given in terms of coordinates (option GUIDING-POINT) or as a point infinitely far away along any of the global axes, either in positive or negative direction. This guiding point determines the orientation of the local y-x- or z-x-plane so that the guiding point is located on the positive local y- or z-side, respectively, of the element; see Figure 5.19. This is valid for the subsequently selected elements.
- A plane (option PLANE) is defined by referring to three nodes. The local coordinate system is defined for all elements in this plane as follows: the local y-x- or z-x-plane, depending on the option chosen, will be parallel with the plane defined by the three nodes. The direction of the local z- or y-axis, correspondingly, will be normal to the plane and with its positive direction according to the right hand rule and the positive rotation defined by the three selected nodes. See Figure 5.20.

Note that if a local coordinate system is defined and one, or both of the nodes of an element are repositioned, then the local coordinate system may become erroneous (the local x-axis may change while the local y- and z-axes are fixed). This may also happen when introducing eccentricities (by the PROPERTY

ECCENTRICITY or PROPERTY GAP commands). In such cases the local coordinate system must be redefined by the user.

Elements for which a local coordinate system is not explicitly defined will have a program calculated (default) local coordinate system. The default local coordinate system is so that the local z-x-plane is either parallel to the global y-axis or the global z-axis depending on whether the element itself is parallel with the global z-axis or not.

The calculated (default) local coordinate system has the same effect as giving the command:

```
PROPERTY LOCAL-COORDINATE ZX-PLANE +Z-GLOBAL-INFINITY
```

for all elements not parallel with the global z-axis, and giving the command:

```
PROPERTY LOCAL-COORDINATE ZX-PLANE +Y-GLOBAL-INFINITY
```

for all elements parallel with the global z-axis.

The local coordinate systems of the elements are printed by the PRINT LOCAL-COORDINATE command and displayed by the LABEL LOCAL-COORDINATE command.

The last column of the print table giving the local coordinate systems for the different elements indicates whether the local coordinate systems are specified (SPEC.) or calculated (CALC.).

PARAMETERS:

YX-PLANE ZX-PLANE	Specifies whether the local y-x- or z-x-plane is to be defined.
+X-GLOBAL-INFINITY	The local y-x- or z-x-plane is defined by the element axis and a guiding point. +/-X, +/-Y, +/-Z specifies a guiding point at an infinite distance along the positive/negative global x-, y- or z-axis respectively.
-X-GLOBAL-INFINITY	See above.
+Y-GLOBAL-INFINITY	See above.
-Y-GLOBAL-INFINITY	See above.
+Z-GLOBAL-INFINITY	See above.
-Z-GLOBAL-INFINITY	See above.
GUIDING-POINT	Specifies that guiding point is to be defined by the user.
gx gy gz	The vector pointing from end 1 of the element to the guiding point. The vector is given in the global coordinate system. See Figure 5.19.
select-elements	Select elements; see Figure 5.3.
PLANE	The local y-x- or z-x-plane is defined by a plane defined by three nodes; see above for an explanation of the method.

node1 node2 node3

The three nodes defining the plane.

Figure 5.19 shows the local coordinate system defined by a guiding point defining the local z-x-plane. The local y-axis is normal to the plane defined by the element axis and the guiding point.

Figure 5.20 shows the local coordinate system defined by using the PLANE option to define the local y-x-plane. The local z-axis is normal to the plane with a direction defined by a right hand rule and a positive rotation through the three nodes.

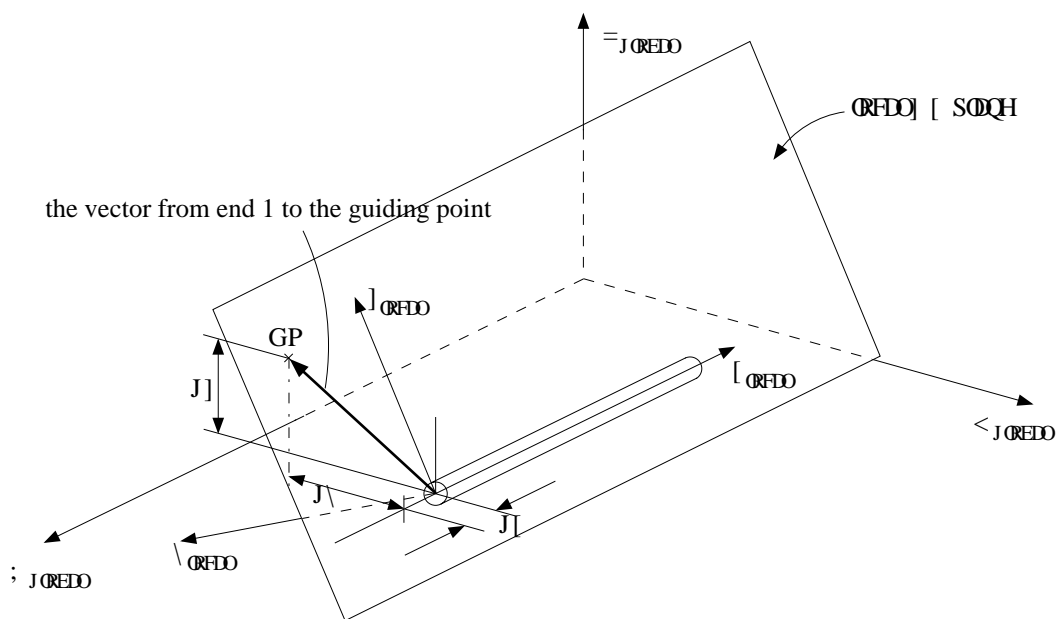


Figure 5.19 Local coordinate system defined by a guiding point defining the local z-x-plane

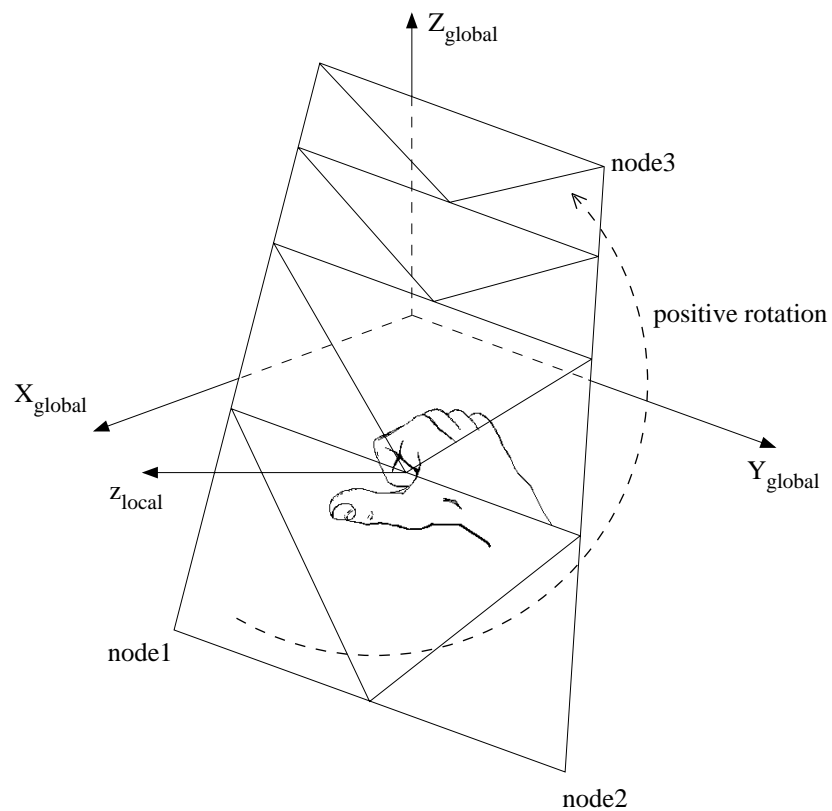


Figure 5.20 Local coordinate system defined by using the PLANE option to define the local y-x-plane

PROPERTY MATERIAL

...	MATERIAL	matno	AXIAL-DAMPER	...
			AXIAL-SPRING	
			DAMPER-TO-GROUND	
			GENERAL-SPRING	
			LINEAR-ELASTIC	
			SHIM-MEMBER	
			SPRING-TO-GROUND	

PURPOSE:

The command defines material data. There material types are:

- axial damper constant,
- axial spring constant,
- damper to ground matrix,
- general spring stiffness matrix,
- linear elastic material data,
- transverse stiffness (spring constant),
- springs to ground matrix.

The linear elastic material is relevant for the beam, truss and non-structural beam elements, the other material types belong to the corresponding element types.

PROPERTY MATERIAL matno AXIAL-DAMPER / AXIAL-SPRING

...	matno	AXIAL-DAMPER	damp
		AXIAL-SPRING	spring

PURPOSE:

The command defines an axial damper / axial spring material type.

The axial damping constant corresponds to the force to be applied in order to obtain a unit velocity in the direction of the basic element. This facility is used in connection with dynamic analysis.

The axial spring constant corresponds to the force to be applied in order to obtain a unit displacement in the direction of the basic element.

PARAMETERS

matno	Material reference number.
damp	Axial damping constant.
spring	Axial spring constant.

PROPERTY MATERIAL **matno** DAMPER-TO-GROUND

...	matno	DAMPER-TO-GROUND	ndof	c _{1,1}	c _{2,1}	...	c _{ndof,ndof}
-----	-------	------------------	------	------------------	------------------	-----	------------------------

PURPOSE:

The command defines a damper to ground material type. The number of d.o.f.s of the damping matrix must correspond to the number of d.o.f.s of the nodes of the relevant damper to ground elements.

The damper to ground matrix is the viscous damping matrix.

The matrix values are given in the local coordinate system of the damper to ground element. The symbol of the damper to ground element on the display indicates its positive x-axis.

PARAMETERS:

matno	Material reference number.
ndof	Number of d.o.f.s
c _{1,1} c _{2,1} ... c _{ndof,ndof}	The elements of the lower triangle of the viscous damping matrix. The elements are given column by column. The elements outside the diagonal will have default value 0.0.

PROPERTY MATERIAL **matno** GENERAL-SPRING

...	matno	GENERAL-SPRING	ndof1	ndof2	$k_{1,1}$	$k_{2,1}$...	$k_{ndof,ndof}$
-----	-------	----------------	-------	-------	-----------	-----------	-----	-----------------

PURPOSE:

The command defines a general-spring stiffness matrix. The number of d.o.f.s of the matrix must correspond to the number of d.o.f.s of the nodes of the relevant general spring elements.

The matrix values are given in the local coordinate system of the general spring element.

PARAMETERS:

matno	Material reference number.
ndof1	Number of d.o.f.s in local node 1.
ndof2	Number of d.o.f.s in local node 2.
$k_{1,1} \ k_{2,1} \dots k_{ndof,ndof}$	The elements of the lower triangle of the stiffness matrix given column by column. The elements outside the diagonal will have default value 0.0. (ndof = ndof1 + ndof2)

NOTES:

Note that the off-diagonal elements (idof,ndof1+idof) corresponding to the translational d.o.f.s (idof is a translational d.o.f.) should be given a negative value when defining a complete diagonal general spring matrix with same number of d.o.f.s in each node:

$$-k_{idof,ndof1+idof} = k_{idof,idof} = k_{ndof1+idof,ndof1+idof}$$

PROPERTY MATERIAL matno LINEAR-ELASTIC

...	matno	LINEAR-ELASTIC	young	poiss	rho	damp	alpha
-----	-------	----------------	-------	-------	-----	------	-------

PURPOSE:

The command defines a linear elastic material type relevant for beam, truss and non-structural beam elements.

PARAMETERS:

matno	Material reference number.
young	Young's modulus.
poiss	Poisson's ratio.
rho	Density.
damp	Specific damping.
alpha	Thermal expansion coefficient.

PROPERTY MATERIAL **matno** SHIM-MEMBER

...	matno	SHIM-MEMBER	ndof1	ndof2	trans-stiff
-----	-------	-------------	-------	-------	-------------

PURPOSE:

The command defines the transverse stiffness of a shim element. The number of d.o.f.s of the matrix must correspond to the number of d.o.f.s of the nodes of the relevant shim elements.

The transverse stiffness is given in relation to the z-axis of the shim element's local coordinate system, i.e. a stiffness in the local x- and y-directions.

PARAMETERS:

matno	Material reference number.
ndof1 ndof2	Number of d.o.f.s in local node 1 and node 2.
trans-stiff	Transverse stiffness normal to the local z-axis.

PROPERTY MATERIAL **matno** SPRING-TO-GROUND

...	matno	SPRING-TO-GROUND	STIFFNESS	ndof	$k_{1,1}$	$k_{2,1}$...	$k_{ndof,ndof}$
			FLEXIBILITY					

PURPOSE:

The command defines a spring to ground material type. The matrix may either be given as a stiffness matrix or a flexibility matrix, the flexibility matrix must be invertible. The number of d.o.f.s of the matrix must correspond to the number of d.o.f.s of the nodes of the relevant spring to ground elements.

The element $k_{i,j}$ of the stiffness matrix corresponds to the force to be given in the i'th d.o.f. to get a unit displacement in the j'th d.o.f.

The element $k_{i,j}$ of the flexibility matrix corresponds to the displacement in the i'th d.o.f. when given a unit force in the j'th d.o.f.

The matrix values are given in the spring to ground element's local coordinate system. The symbol of the spring to ground element on the display indicates its positive x-axis.

The stiffness and flexibility matrices are symmetric about the diagonal and are thus specified by only giving the lower triangle of the matrices.

$$\mathbf{F} = \mathbf{K} \mathbf{r}$$

$$\mathbf{r} = \mathbf{K}^{-1} \mathbf{F}$$

where:

\mathbf{F} is the force vector

\mathbf{r} is the displacement vector

\mathbf{K} is the stiffness matrix

\mathbf{K}^{-1} is the flexibility matrix

PARAMETERS:

matno Material reference number.

ndof Number of d.o.f.s

$k_{1,1} \ k_{2,1} \dots k_{ndof,ndof}$ The elements of the lower triangle of the stiffness or flexibility matrix. The elements are given column by column ($k_{row,column}$). The elements outside the diagonal will have default value 0.0.

PROPERTY SECTION

...	SECTION	sctno	BAR	...
			BOX	
			CHANNEL	
			DOUBLE-BOTTOM	
			GENERAL	
			I	
			L	
			PIPE	
			UNSYM.-I	

PURPOSE:

The command defines geometrical data for the cross sections. The section types are:

- bar,
- box,
- channel,
- double-bottom,
- general,
- I (or H),
- L (angle)
- pipe,
- un-symmetrical I.

Cross sectional data are relevant for beam, truss and non-structural beam elements. In the case of truss elements the data is only used to calculate the cross sectional area.

Figures accompanying the description of the cross sections explain the parameters defining the sections and show the local coordinate system of each section type. The local x-axis is directed into the paper plane. The effect of rounded corners are not taken into account when calculating the sectional properties (the torsional moment of inertia).

Note that the cross sections should be used with care. The dimensions of a cross section should not be given so that the specified cross section degenerates into another type. The formulae used to calculate the cross sectional properties are based on the assumption that the cross sections have a reasonable shape. For all

standard cross section types the formulae are sufficiently correct. However, a study has shown that the formulae for the torsional moments of inertia are most sensitive to misuse of the cross sections. The shear centre location z-component for the un-symmetrical I and L sections and the shear area in the direction of y-axis for the I and un-symmetrical I sections are also sensitive to misuse of the cross sections.

Appendix B 1 describes in detail how the moments of inertia, shear areas, shear centres, etc. are calculated.

PROPERTY SECTION sctno BAR

...	sctno	BAR	hz	bb	bt	sfy	sfz
-----	-------	-----	----	----	----	-----	-----

PURPOSE:

The command defines a bar cross section.

PARAMETERS:

sctn Section reference number

hz Height

bb Width at bottom

bt Width at top

sfy sfz Factors modifying the shear areas calculated by the program. The modified shear areas are (see the PRINT SECTION command for an explanation of the parameters):

$$SHARY_{\text{modified}} = SHARY_{\text{program}} \times sfy$$

$$SHARZ_{\text{modified}} = SHARZ_{\text{program}} \times sfz$$

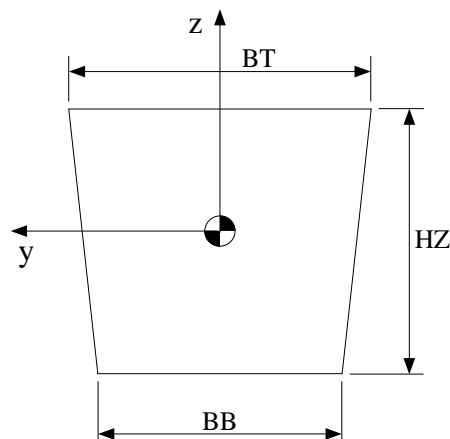


Figure 5.21 Bar section

PROPERTY SECTION **sctno BOX**

...	sctno	BOX	hz	by	tt	ty	tb	sfy	sfz
-----	-------	-----	----	----	----	----	----	-----	-----

PURPOSE:

The command defines a box cross section.

PARAMETERS:

sctn	Section reference number
hz	Height
by	Width
tt	Thickness of top flange
ty	Thickness of webs (vertical walls)
tb	Thickness of bottom flange
sfy sfz	Factors modifying the shear areas calculated by the program. The modified shear areas are (see the PRINT SECTION command for an explanation of the parameters): $SHARY_{\text{modified}} = SHARY_{\text{program}} \times sfy$ $SHARZ_{\text{modified}} = SHARZ_{\text{program}} \times sfz$

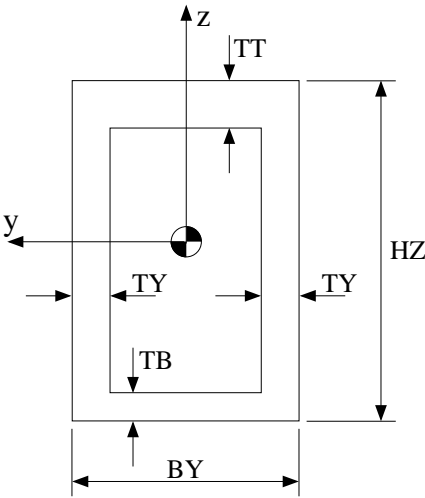


Figure 5.22 Box section

PROPERTY SECTION **sctno** CHANNEL

...	sctno	CHANNEL	hz	by	tz	ty	sfy	sfz	POSITIVE
									NEGATIVE

PURPOSE:

The command defines a channel cross section.

PARAMETERS:

sctn	Section reference number
hz	Height
by	Width of top and bottom flanges
tz	Thickness of top and bottom flanges
ty	Thickness of web
sfy sfz	Factors modifying the shear areas calculated by the program. The modified shear areas are (see the PRINT SECTION command for an explanation of the parameters): $SHARY_{\text{modified}} = SHARY_{\text{program}} \times sfy$ $SHARZ_{\text{modified}} = SHARZ_{\text{program}} \times sfz$
POSITIVE	Web location in the positive local y-direction
NEGATIVE	Web location in the negative local y-direction

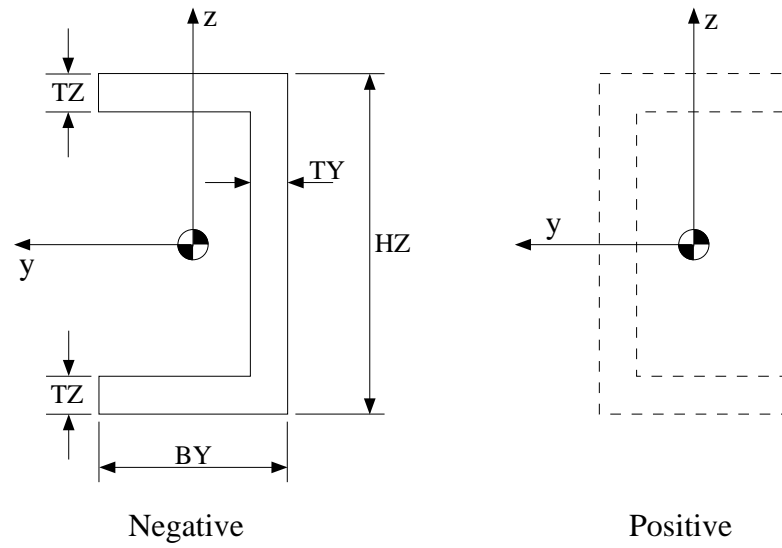


Figure 5.23 Channel section

PROPERTY SECTION **sctno** DOUBLE-BOTTOM

...	sctno	DOUBLE-BOTTOM	hz	ty	tb	tt	by	sfy	sfz
-----	-------	---------------	----	----	----	----	----	-----	-----

PURPOSE:

The command defines a double-bottom type cross section.

PARAMETERS:

sctno	Section reference number
hz	Height
ty	Thickness of web
tb	Thickness of bottom plate
tt	Thickness of top plate
by	Effective width of plates
sfy sfz	Factors modifying the shear areas calculated by the program. The modified shear areas are (see the PRINT SECTION command for an explanation of the parameters):
	$SHARY_{\text{modified}} = SHARY_{\text{program}} \times sfy$
	$SHARZ_{\text{modified}} = SHARZ_{\text{program}} \times sfz$

NOTES:

The user should use the double-bottom section with care. This is because the sectional properties are calculated in the same way as for the symmetrical I section. Only the torsional moment of inertia is increased to take into account the shear flow along the top and bottom plates. The effects in other directions are not considered. The section may be used for estimating type of analyses.

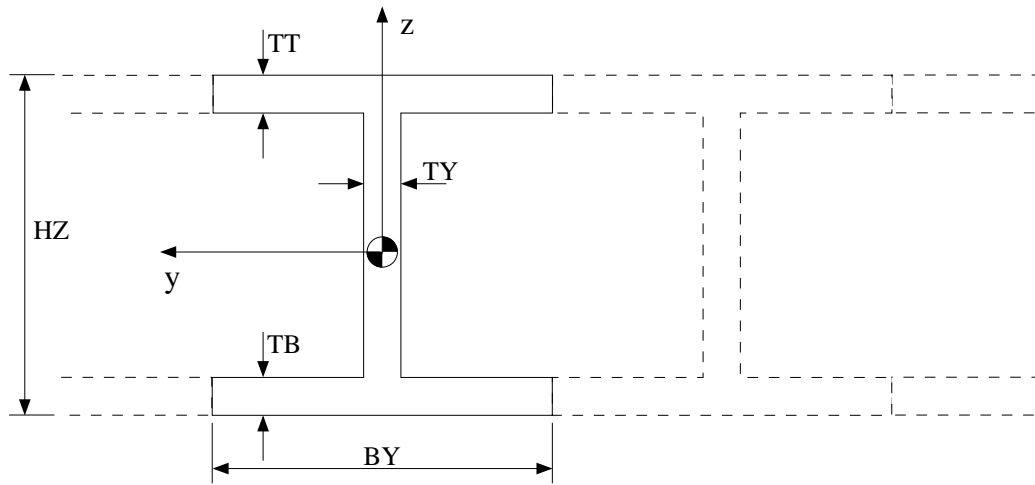


Figure 5.24 Double-bottom section

PROPERTY SECTION **sctno** GENERAL

...	sctno	GENERAL	area	ix	iy	iz	iyz	wxmin	wymin	wzmin	shary	...
...	sharz	shceny	shcenz	sy	sz							

PURPOSE:

The command defines a general section. All sectional data are defined directly. The following should be noted:

- For beams the area and moments of inertia are required while only the area is required for trusses.
- The product of inertia (I_{YZ}) is zero for all bi-symmetrical sections.
- The minimum sectional moduli (W_{Xmin} , W_{Ymin} and W_{Zmin}) are required by the FRAMEWORK post-processor for calculating stresses in the sections.
- Shear deformations will not be accounted for if the shear areas are zero.
- The shear centre location must be specified if the shear centre does not coincide with the element axis.
- The static area moments are used in connection with un-symmetrical sections (IYZ0) to re-compute section values to the principal axis.

PARAMETERS:

sctno	Section reference number
area	Cross sectional area (>0.0)
ix	Torsional moment of inertia about shear centre (>0.0)
iy	Moment of inertia about y-axis (>0.0)
iz	Moment of inertia about z-axis (>0.0)
iyz	Product of inertia about y- and z-axes
wxmin	Minimum torsional sectional modulus about shear centre (≥ 0.0)
wymin	Minimum sectional modulus about y-axis (≥ 0.0)
wzmin	Minimum sectional modulus about z-axis (≥ 0.0)
shary	Shear area in the direction of y-axis (≥ 0.0)
sharz	Shear area in the direction of z-axis (≥ 0.0)
shceny	Shear centre location from centroid y-component

shcenz	Shear centre location from centroid z-component
sy	Static area moment about y-axis (≥ 0.0)
sz	Static area moment about z-axis (≥ 0.0)

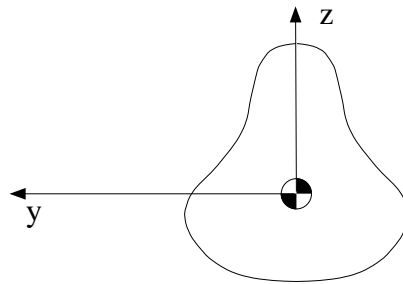


Figure 5.25 General section

PROPERTY SECTION **sctno I**

...	sctno	I	hz	bt	tt	ty	bb	tb	sfy	sfz
-----	-------	---	----	----	----	----	----	----	-----	-----

PURPOSE:

The command defines a symmetrical I (or H) cross section.

PARAMETERS:

sctn Section reference number

hz Height

bt Width of top flange

tt Thickness of top flange

ty Thickness of web

bb Width of bottom flange

tb Thickness of bottom flange

sfy sfz Factors modifying the shear areas calculated by the program. The modified shear areas are (see the PRINT SECTION command for an explanation of the parameters):

$$SHARY_{\text{modified}} = SHARY_{\text{program}} \times sfy$$

$$SHARZ_{\text{modified}} = SHARZ_{\text{program}} \times sfz$$

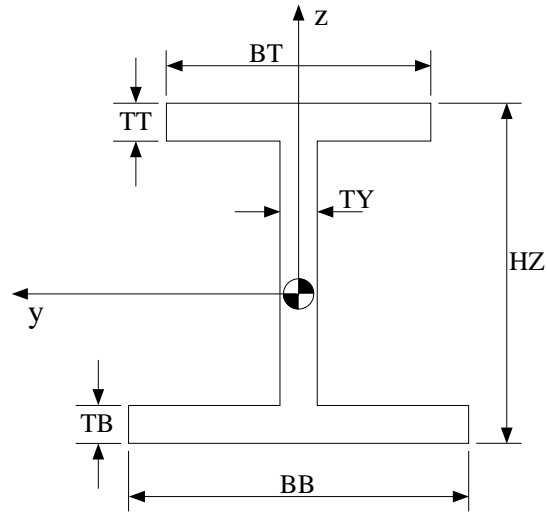


Figure 5.26 Symmetrical I (or H) section

PROPERTY SECTION **sctno L**

...	sctno	L	hz	ty	by	tz	sfy	sfz	POSITIVE
									NEGATIVE

PURPOSE:

The command defines an L cross section.

PARAMETERS:

sctn	Section reference number
hz	Height
ty	Thickness of web
by	Width of flange
tz	Thickness of flange
sfy sfz	Factors modifying the shear areas calculated by the program. The modified shear areas are (see the PRINT SECTION command for an explanation of the parameters): $SHARY_{\text{modified}} = SHARY_{\text{program}} \times sfy$ $SHARZ_{\text{modified}} = SHARZ_{\text{program}} \times sfz$
POSITIVE NEGATIVE	Web location in the local y-direction

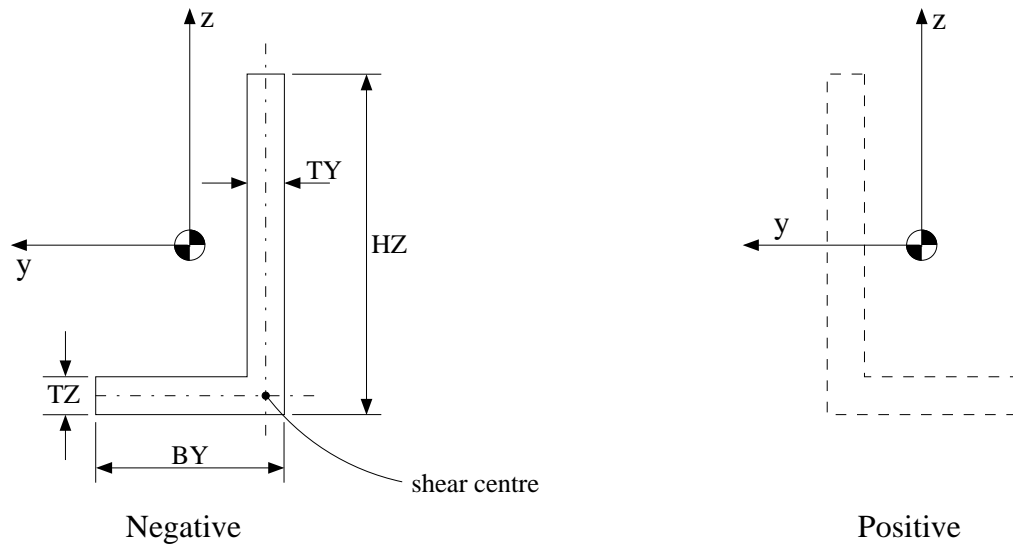


Figure 5.27 L section

PROPERTY SECTION sctno PIPE

...	sctno	PIPE	dy	t	sfy	sfz
-----	-------	------	----	---	-----	-----

PURPOSE:

The command defines a pipe cross section.

PARAMETERS:

sctn Section reference number

dy Outer diameter

t Thickness of wall

sfy sfz Factors modifying the shear areas calculated by the program. The modified shear areas are (see the PRINT SECTION command for an explanation of the parameters):

$$SHARY_{\text{modified}} = SHARY_{\text{program}} \times sfy$$

$$SHARZ_{\text{modified}} = SHARZ_{\text{program}} \times sfz$$

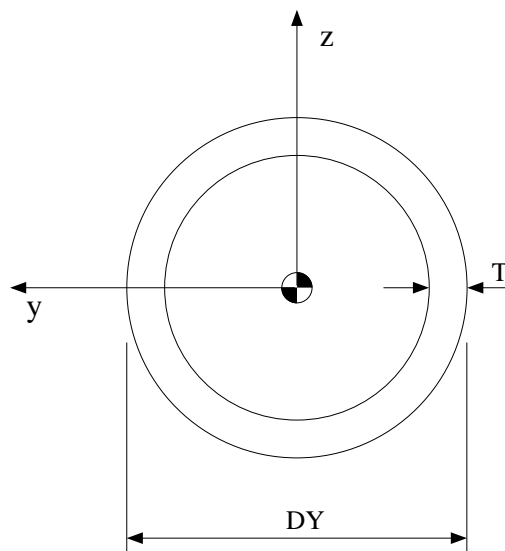


Figure 5.28 Pipe section

PROPERTY SECTION sctno UNSYM.-I

...	sctno	UNSYM.-I	hz	bt	b1	tt	ty	bb	b2	tb	sfy	sfz
-----	-------	----------	----	----	----	----	----	----	----	----	-----	-----

PURPOSE:

The command defines an un-symmetrical I cross section.

PARAMETERS:

sctn	Section reference number
hz	Height
bt	Width of top flange
b1	Width of part of top flange along positive y-axis
tt	Thickness of top flange
ty	Thickness of web
bb	Width of bottom flange
b2	Width of part of bottom flange along positive y-axis
tb	Thickness of bottom flange
sfy sfz	Factors modifying the shear areas calculated by the program. The modified shear areas are (see the PRINT SECTION command for an explanation of the parameters): <div> $SHARY_{\text{modified}} = SHARY_{\text{program}} \times sfy$ $SHARZ_{\text{modified}} = SHARZ_{\text{program}} \times sfz$ </div>

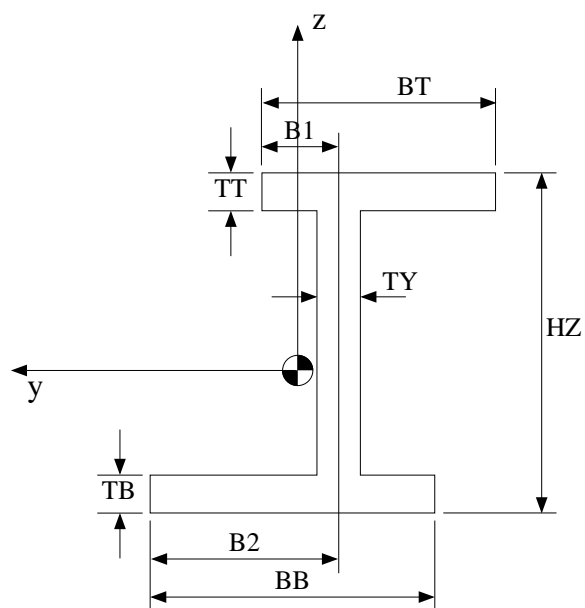


Figure 5.29 Un-symmetrical I section

PROPERTY SOIL

...	SOIL	SAND	num	gamtot	phi			ocr		open	r/p-rat	tzzres
		CLAY	num	gamtot	suz0	suz100	epsc	ocr	api-j			

PURPOSE:

The command defines the soil types to be referred to from the soil profile (layer divisions). The command is split into definition of SAND or CLAY.

PARAMETERS:

SAND	Define the soil type (number) as sand.
CLAY	Define the soil type (number) as clay.
num	Soil type (number).
gamtot	Total unit weight.
phi	Angle of internal friction (degrees).
suz0	Un-drained shear strength at z=0.
suz100	Un-drained shear strength at z=100.
epsc	Strain at half of maximum stress.
ocr	Over-consolidation ratio.
api-j	J factor for API code.
open	Code for open gap (0 or 1).
r/p-rat	Residual / peak skin friction ratio.
tzzres	T-Z curve z displacement from peak to residual skin friction.

NOTES:

This command may also be used to modify existing data.

See also the Gensod User Manual for specific soil related explanation.

RE-DISPLAY

RE-DISPLAY

PURPOSE:

The command displays the same picture (apart from labels) as the last DISPLAY command.

READ

READ	prefix	filnam	sup-el-no
------	--------	--------	-----------

PURPOSE:

The command reads an Input Interface File describing a FE model into Preframe. Provided no errors occur during reading, the model may be modified by commands available in Preframe and an updated Input Interface File may be generated by the WRITE command.

The Input Interface File, which must be a first level superelement, may have been generated by preprocessors other than Preframe, Preframe will still be able to print and display data.

Using the READ command involves that the current model (and command log) file will be closed and a new opened. The name of the new model (and command log) file is according to the input given in the READ command.

The READ command involves reading of the file:

prefixTsup-el-no.FEM

The Input Interface File is read three times. Each time the following message is written (n=1,2,3):

READING PASS n

When the reading has been completed a message giving the number of nodes, elements and load cases is printed, e.g.:

```
83  NODES READ
167 ELEMENTS READ
5   LOADCASES READ
```

PARAMETERS:

prefix	General file prefix
filnam	Model and command log file name
sup-el-no	Superelement number

NOTES:

It is not possible to read superelements containing element type no. 70 (higher level data), i.e. AMATRIX record where all relevant data are stored as stiffness, mass, damping matrices a.s.o. (ref. AMD ... records).

Preframe can only read one BGRAV (gravity) card for each load case.

See also note in connection with the command SET COMMAND-INPUT-FILE.

RENUMBER

RENUMBER	ELEMENT	oldnumber	newnumber
	LOADCASE		
	MATERIAL		
	NODE		
	SECTION		

PURPOSE:

The command changes the number of previously created nodes and elements, as well as the reference number of previously defined load cases, materials and cross sections.

PARAMETERS:

oldnumber	Number to be changed
newnumber	The new number

ROTATE

ROTATE	X-AXIS	degrees
	Y-AXIS	
	Z-AXIS	

PURPOSE:

The command rotates the picture displayed with the DISPLAY command or plotted with the PLOT command. The rotations are about the global axis.

The rotations are accumulative and will be used for all subsequent DISPLAY and PLOT commands.

PARAMETERS:

degrees Angle in degrees

SET

SET	ALIGNMENT-AUTOMATIC	...
	ANGLE-TOLERANCE	angtol
	ASSIGN-OPTION	...
	COMMAND-INPUT-FILE	...
	COORDINATE-TOLERANCE	cotol
	DEFAULT	...
	DISPLAY-GAP	display-gap-value
	ELEMENT-LOAD-DISTANCE-MODE	...
	GRAPHICS	...
	JOURNALLING	...
	LEGAL-GAP	legal-gap-value
	MODEL-FILE	...
	NUMBERING-AUTOMATIC	...
	PLOT	...
	PRINT	...
	SOIL-PROFILE-X-Y	...
	CAN-STUB-LENGTH-PARAMETER	...
	UNIT-VECTOR-TOLERANCE	...
	WRITE-MODE	...
	ZERO-GAP	zero-gap-value

PURPOSE:

The command sets different parameters for controlling the execution of the other commands. Some of the options are explained below, some are treated in more detail in the following.

- The ANGLE-TOLERANCE option specifies the angle tolerance used for determining whether an angle is 90 degrees or not. The angle tolerance is given in degrees and the default value is 1/1000.
- The COORDINATE-TOLERANCE option specifies the coordinate tolerance used for deciding whether two points (nodes) have the same location or not and for deciding whether a node lies in a plane or on a line. The coordinate tolerance is given in the same unit as the coordinates. The default value is 0.1.

The DISPLAY-GAP, LEGAL-GAP and ZERO-GAP options are used in the DISPLAY FOOTPRINT command. See Section 3.16 for more details.

- The DISPLAY-GAP option sets a new value for the display-gap-value.
- The LEGAL-GAP option sets new values for the legal-gap-value, display-gap-value and zero-gap-value. Any values given for the DISPLAY-GAP and ZERO-GAP options are overridden. The display-gap-value will become twice the legal-gap-value, and the zero-gap-value will become the legal-gap-value/100.
- The ZERO-GAP option sets a new value for the zero-gap-value.

PARAMETERS:

angtol	Angle tolerance
cotol	Coordinate tolerance
display-gap-value	The value below which gaps are displayed (marked with a line) in the DISPLAY FOOTPRINT command. Gaps larger than the specified value will not be shown.
legal-gap-value	The minimum allowable gap between brace-chord intersections.
zero-gap-value	The gap size below which an overlap is assumed.

SET ALIGNMENT-AUTOMATIC

...	ALIGNMENT-AUTOMATIC	ON
		OFF

PURPOSE:

The command switches on and off automatic alignment of elements created. When set to ON, elements created by the GENERATE command (i.e. jacket structure, line of elements, T-brace, K-brace and X-brace) and the SPLIT command will automatically be assigned alignment attributes.

PARAMETERS:

ON	The automatic alignment is switched on.
OFF	The automatic alignment is switched off. This is the default condition.

NOTES:

If the node in one end of aligned elements moves then all intermediate nodes will be moved to keep the elements on a straight line. Also, if the first or last element is given an eccentricity then all intermediate elements will be given necessary eccentricities to keep the elements on a straight line.

SET ASSIGN-OPTION

...	ASSIGN-OPTION	MANUAL-NUMBERING	...	ON
		SECTION-NUMBER		OFF

PURPOSE:

The command is used to switch ON / OFF how to handle input alternatives regarding node/element and section numbers in connection with the ASSIGN CAN / STUB / CONE / SEGMENT command. The user may switch these options ON and OFF during the session.

PARAMETERS:

MANUAL-NUMBERING	Node and element number for created node and element given by user.
SECTION-NUMBER	Give section number instead of PIPE section geometry data to be connected to created element.
ON	Switch option on.
OFF	Switch option off.

NOTES:

.See also Section 3.6.1 regarding the ASSIGN-OPTION switches.

By default the options are OFF.

SET CAN-STUB-LENGTH-PARAMETER

...	CAN-STUB-LENGTH-PARAMETER	canfac	canmin	stubfac	stubmin
-----	---------------------------	--------	--------	---------	---------

PURPOSE:

The command sets the default lengths used when calculating can and stub lengths in tubular joints. The default can and stub lengths for tubular joints are calculated according to pre-defined geometric rules. For typical values see Figure 3.11.

PARAMETERS:

canfac	Free can length as fraction of can diameter (default = 1/4 of Diameter of chord / can).
canmin	Minimum free can length (default = 0.3 m).
stubfac	Free stub length as fraction of stub diameter (default = diameter of stub / brace).
stubmin	Minimum free stub length (default = 0.6 m).

NOTES:

See also:

ASSIGN CAN ...
 ASSIGN STUB ...
 SPLIT ...

SET COMMAND-INPUT-FILE

...	COMMAND-INPUT-FILE	filnam
-----	--------------------	--------

PURPOSE:

The command opens the specified file as a command input file. The # command will read commands into Preframe.

See Section 4.1.7 for a description of how Preframe conveniently may be run in batch using the SET COMMAND-INPUT-FILE command combined with the # command.

The command input file cannot have the same name as the Preframe command log file.

PARAMETERS:

filnam	Name of the command input file, the name is given without the file extension (which is required to be JNL)
--------	--

NOTES:

If the command input file opened by this command and executed by the # command contains the READ command, execution of the command input file will be aborted after executing the READ command.

SET DEFAULT

...	DEFAULT	MATERIAL	matno
			NONE
		SECTION	sctno
			NONE

PURPOSE:

The command defines material and/or section numbers to be automatically assigned (connected) to elements subsequently created.

The command may conveniently be used as follows: Prior to creating a number of elements having the same section (and/or material) define the appropriate section as default. Then switch to a new default section prior to creating several new elements having another section. This command spares you from assigning (connecting) the various sections afterwards. The assignments may be overwritten by explicit assignments (PROPERTY CONNECT) afterwards.

PARAMETERS:

matno Material reference number

sctno Section reference number

NONE No material/section numbers will be assigned. This is the default condition for both section and material.

SET ELEMENT-LOAD-DISTANCE-MODE

...	ELEMENT-LOAD-DISTANCE-MODE	PROJECTION-OF-NODES-ON-ELEMENT-AXIS
		END-OF-FLEXIBLE-PART-OF-ELEMENT

PURPOSE:

The command decides whether the extent of distributed element loads are relative to the points where the nodes are projected onto the element axis, or from the ends of the flexible part of the element. This choice has effect only for elements with eccentricities.

The following example will illustrate the effect of the two options. Figure 5.30 shows a beam element connected to nodes A and B. Eccentricities are defined in both ends: e_A from node A to beam end 1, and e_B from node B to beam end 2.

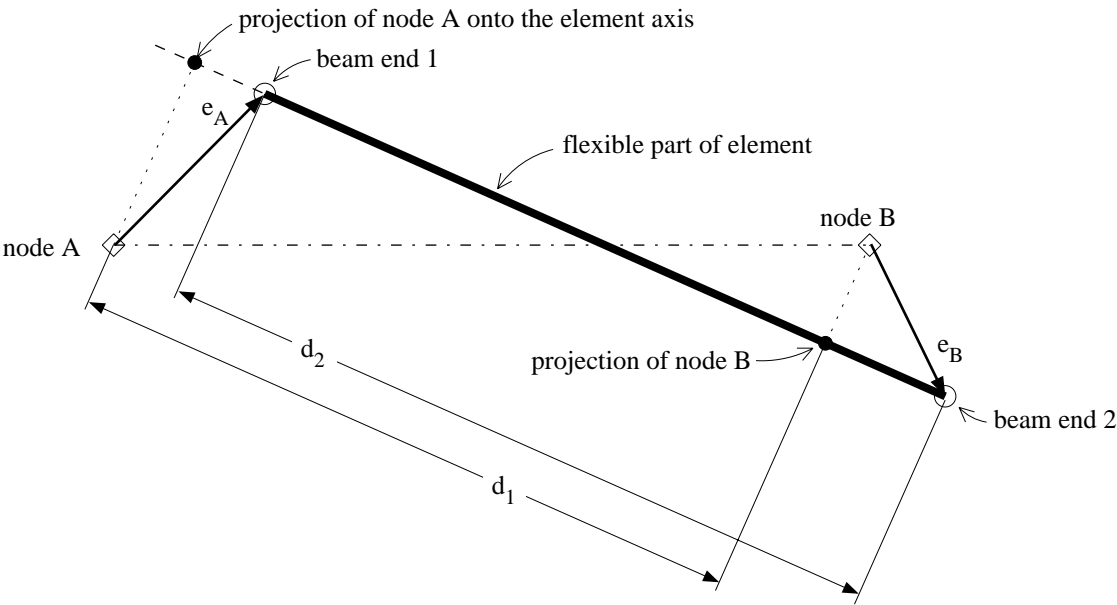


Figure 5.30 Effect of ELEMENT-LOAD-DISTANCE-MODE command

The PROJECTION-OF-NODES-ON-ELEMENT-AXIS option will give a load referring to the distance d_1 while the END-OF-FLEXIBLE-PART-OF-ELEMENT will give a load referring to the distance d_2 . The former option is the default.

PARAMETERS:

PROJECTION-OF-NODES-ON-ELEMENT-AXIS	Distance will be from the projection of nodes onto the element axis, the default choice.
-------------------------------------	--

END-OF-FLEXIBLE-PART-OF-ELEMENT

Distance will be from the ends of the flexible part of the element.

SET GRAPHICS

...	GRAPHICS	ALTERNATIVE-SCREEN-DEVICE	...
		AUTO	
		BASIC-ELEMENT-MODE	
		CHARACTER-TYPE	
		COLOUR	
		DEVICE	
		EYE-DIRECTION	
		HIDDEN	
		INPUT	
		PLOT-FILE	
		PRESENTATION	
		SHRINK-FACTOR	
		SIZE-SYMBOLS	

PURPOSE:

Enables the user to set different parameters which control the DISPLAY, PLOT and LABEL commands.

PARAMETERS:

ALTERNATIVE-SCREEN-DEVICE This option is presently not in use.

BASIC-ELEMENT-MODE This option is presently not in use.

SET GRAPHICS AUTO

...	AUTO	ON
		OFF

PURPOSE:

The command involves that updates to the model will result in a refresh of the display with the updates included. This is the default mode when the graphical user interface has been selected.

PARAMETERS:

ON Updates to the model will automatically be drawn.

OFF Option turned off.

SET GRAPHICS CHARACTER-TYPE

...	CHARACTER-TYPE	SOFTWARE
		HARDWARE

PURPOSE:

The command decides whether characters output on the graphic device are drawn (SOFTWARE) or device generated (HARDWARE). Device generated characters are much faster to produce, but there may be restrictions on the character size and orientation. By default, characters are drawn (SOFTWARE).

PARAMETERS:

SOFTWARE	Drawn characters are used.
HARDWARE	Device generated characters are used.

SET GRAPHICS COLOUR

...	COLOUR	BOUNDARY-CONDITION	...	WHITE	...	DARK
		ELEMENT-NUMBERS		GRAY		
		ELEMENTS		BLACK		
		LOAD-ARROWS		BLUE		LIGHT
		MATERIAL-NUMBERS		GREEN		
		NODE-NUMBERS		ORANGE		
		NODE-SYMBOLS		RED		MEDIUM
		ORIGIN-SYMBOL		VIOLET		
		SECTION-NUMBERS		YELLOW		
		SUPER-NODE-SYMBOL				

PURPOSE:

The command changes the colours used in display and on plots.

PARAMETERS:

self-explanatory

NOTES:

DARK / LIGHT / MEDIUM option not available for BLACK and WHITE.

SET GRAPHICS DEVICE

...	DEVICE	CDC-721
		TX4014-15-16-54
		TX4105
		TX4107-09-13-15
		X-WINDOW
		WINDOWS
		DUMMY

PURPOSE:

The command specifies the type of graphics device for output of pictures by the DISPLAY command. The alternatives varies with the hardware used. The default devices are WINDOWS and X-WINDOW for PC and Unix respectively.

SET GRAPHICS EYE-DIRECTION

...	EYE-DIRECTION	eyex	eyey	eyez
-----	---------------	------	------	------

PURPOSE:

The command specifies the viewpoint used for displaying the model. The same effect can be achieved by the ROTATE command.

PARAMETERS:

eyex	x-coordinate of the viewpoint (the eye)
eyey	y-coordinate of the viewpoint (the eye)
eyez	z-coordinate of the viewpoint (the eye)

SET GRAPHICS HIDDEN

...	HIDDEN	ON
		OFF

PURPOSE:

The command sets the display mode to hidden. This is relevant only for DISPLAY JOINT and when the SET GRAPHICS PRESENTATION BEAM-ELEMENT FACET command has been used.

SET GRAPHICS INPUT

...	INPUT	ON
		OFF

PURPOSE:

The command switches between graphical user interface and line-mode. A guide to using the graphical user interface is provided in Section 3.1. This command will also switch to 'SET GRAPHICS AUTO ON'.

This command is irrelevant in a Windows NT environment as the graphical user interface is the only option for interactive execution of the program.

SET GRAPHICS PLOT-FILE

...	PLOT-FILE	prefix	filnam
-----	-----------	--------	--------

PURPOSE:

The SET PLOT FILE command has the same functionality as this command, you may want to use that one as it is more consistent with the other SET PLOT commands.

The command sets the name of the plot file. By default the plot file will have the same name as the model and command log files. The extension of the plot file varies with the plot format; see the SET PLOT FORMAT command.

PARAMETERS:

prefix Prefix of the plot file

filnam Name of the plot file

NOTES:

This command closes the current plot file (if such exists) enabling this to be sent to a laser printer without having to exit the execution of Preframe.

SET GRAPHICS PRESENTATION

...	PRESENTATION	BEAM-ELEMENT	FACET
			SILHOUETTE
			WIRE-FRAME
		FILLED-ELEMENT	ON
			OFF
		LOAD	CONNECTED-ARROWS
			NUMERICAL
			VALUED-ARROWS

PURPOSE:

The command sets the draw mode for elements and loads.

PARAMETERS:

BEAM-ELEMENT	Set the draw mode for beam elements.
FACET	Tubular elements (pipe section) will be shown in a faceted mode. Non-tubes will be shown in a solid mode.
SILHOUETTE	Tubular elements (pipe section) will be shown as silhouettes. I.e. the diameter of the pipe is shown. Non-tubes will be shown in a solid mode as for the FACET option.
WIRE-FRAME	All elements are shown as wire frames only. I.e. the neutral axis of the elements are drawn. This is the default option. When viewing the complete model, the WIRE-FRAME draw mode ensures a fast update of the picture.
FILLED-ELEMENT	Switch filling of elements on and off. This command is relevant for beam elements only when these are displayed in SILHOUETTE and FACET mode. (The command is also relevant for 2-D elements read into Preframe by the READ command.)
LOAD	Set the display mode for loads (see the ADD-DISPLAY LOAD command).
CONNECTED-ARROWS	Load values are shown as arrows and with their tails connected.
NUMERICAL	Load values are shown as numerical values.
VALUED-ARROWS	Load values are shown as both arrows and with their tails connected and numerical values.

SET GRAPHICS SHRINK-FACTOR

...	SHRINK-FACTOR	shrinkfac
-----	---------------	-----------

PURPOSE:

The command specifies shrinking of the elements. The effect is illustrated by the example of Figure 5.31 below. The advantage of shrinking basic elements is that each element can be distinguished from the others.

PARAMETERS:

shrinkfac Shrink factor

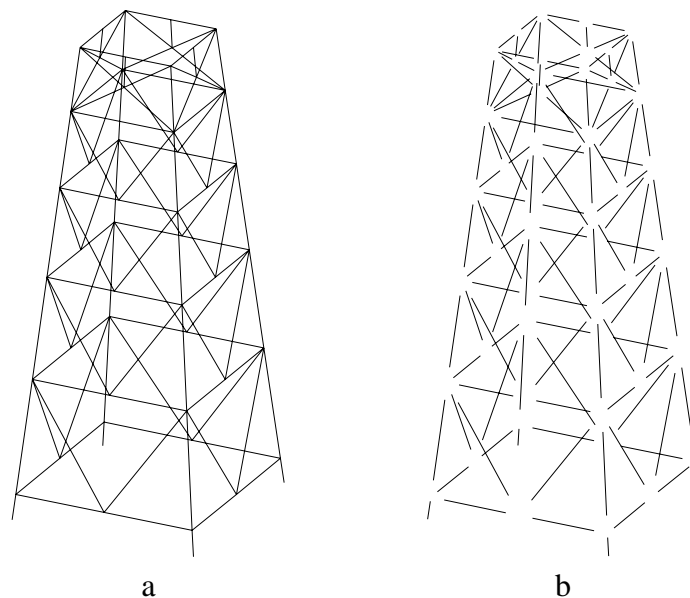


Figure 5.31 a) Normal element mesh, shrink factor = 1.0 b) Shrunken elements, shrink factor = 0.7

SET GRAPHICS SIZE-SYMBOLS

...	SIZE-SYMBOLS	ALL-NUMBERS	size
		BOUNDARY-CONDITION-SYMBOLS	
		ELEMENT-NUMBERS	
		LOAD-ARROWS	
		LOAD-VALUES	
		LOCAL-COORDINATE-SYMBOLS	
		MATERIAL-NUMBERS	
		NODE-NUMBERS	
		NODE-SYMBOLS	
		ONE-NODED-ELEMENT-SYMBOLS	
		ORIGIN-SYMBOL	
		SECTION-NUMBERS	

PURPOSE:

The command specifies the sizes of the symbols appearing on the displayed picture and the plot. The symbol sizes are given in mm.

If the sizes of numbers are changed, the numbers will be drawn with SOFTWARE (see the SET GRAPHICS CHARACTER-TYPE command) characters in order to ensure correct sizes.

PARAMETERS:

size	Symbol size in mm.
ALL-NUMBERS	All numbers, elements, nodes, materials, etc. will be re-sized.
BOUNDARY-CONDITION-SYMBOLS	Symbols for boundary conditions will be re-sized.
ELEMENT-NUMBERS	Element numbers will be re-sized.
LOAD-ARROWS	Load arrows will be re-sized.
LOAD-VALUES	Load values will be re-sized.
LOCAL-COORDINATE-SYMBOLS	Local coordinate symbols will be re-sized.
MATERIAL-NUMBERS	Material numbers will be re-sized.
NODE-NUMBERS	Node numbers will be re-sized.

NODE-SYMBOLS

Symbols for the nodes will be re-sized.

ONE-NODED-ELEMENT-SYMBOLS

Symbols for one node elements will be re-sized. These are the elements connected to only one node, i.e. SPRING-TO-GROUND, and DAMPER-TO-GROUND.

ORIGIN-SYMBOLS

Symbols for the origin will be re-sized.

SECTION-NUMBERS

Section numbers will be re-sized.

SET JOURNALLING

...	JOURNALLING	GRAPHICS	...	ON
		PRINT		OFF

PURPOSE:

The command is used to switch ON / OFF logging onto the journal file graphic / display and print commands. The user may switch these options ON and OFF during the session.

PARAMETERS:

GRAPHICS Graphic commands.

PRINT Print commands.

ON Switch on.

OFF Switch off.

NOTES:

When journalling of graphics command is turned on, the echo from the following commands are put on the journal file: DISPLAY, RE-DISPLAY, ADD-DISPLAY, LABEL, ZOOM, ROTATE and PLOT.

Note that the action from the left quick buttons column (closest to the graphical display area) are not journalled. Hence, only use the right quick buttons column (and the ordinary menu buttons) when input is given from the graphical input screen and graphics journalling is on.

By use of journalling of graphics commands, e.g. a sequence of generated plots may be reproduced by the actual journal file. If PREFRAME is run in line mode, the plots may be generated without any display on the screen if the command SET GRAPHIC DEVICE DUMMY is added to the journal file prior to the first display command.

By default the options are OFF.

SET MODEL-FILE

...	MODEL-FILE	prefix	filnam	NEW	sup-el-no
				OLD	

PURPOSE:

The command closes the current model file and opens another model file without exiting and re-entering the program.

PARAMETERS:

prefix	File name prefix
filnam	File name given without the file extension
NEW	A new model file will be created
OLD	An old model file will be opened
sup-el-n	Superelement number

SET NUMBERING-AUTOMATIC

...	NUMBERING-AUTOMATIC	ON
		OFF
		OLD-SYSTEM
		NEW-SYSTEM

PURPOSE:

The command switches on and off automatic numbering of nodes and elements created. I.e. rather than prompting the user for numbers during creation of nodes and elements the program selects the numbers itself.

The effect of the command will be the same as if the default node/element number is chosen when single nodes/elements are created, and the AUTO option is chosen when several nodes/elements are created. Nodes and element numbers created by a COPY command are based on an increment of 100 for both the node and element numbers.

The program will log commands as if the OFF option is chosen. This means that the node/element numbers or numbering options (STEP or AUTO) chosen by the program during automatic numbering will be logged. This ensures that an identical model will be re-created when using the command log file as input to a future execution.

PARAMETERS:

ON	The automatic numbering is switched on.
OFF	The automatic numbering is switched off. This is the default condition.
OLD-SYSTEM	The automatic numbering is switched to be compatible with input files made prior to Preframe version 6.9-01, see notes below
NEW-SYSTEM	The automatic numbering is switched to the updated numbering system used when switched to ON. This is default from version 6.9-01.

NOTES:

The automatic numbering system for nodes and elements has been updated due to lack of functionality (e.g. highest used node and element numbers were not correctly determined after use of COPY and DELETE commands). To obtain compatibility with 'old models' (i.e. give the same node and element numbers as in previous version of Preframe when running command input files with the automatic numbering activated) the following command should be added at the top of the command input file: SET NUMBERING-AUTOMATIC OLD-SYSTEM END

Hence, if you are concerned with compatibility, search in your command input file for the text (command) AUTO and insert the above command if AUTO is found.

If you continue to work on the model after executing the 'old' command input file it is strongly recommended to switch to the new numbering routines by the command: SET NUMBERING-AUTOMATIC NEW-SYSTEM END

Note: When the command SET NUMBERING-AUTOMATIC ON is given as input, the system logs SET NUMBERING-AUTOMATIC OFF on the journal file and AUTO is used to replace the node and element numbers (when more than one number is required).

SET PLOT

...	PLOT	COLOUR	ON	
			OFF	
		FILE	file-prefix	file-name
		FORMAT	number	
			SESAM-NEUTRAL	
			POSTSCRIPT	
			HPGL-7550	
			HPGL-2	
			CGM-BINARY	
		ORIENTATION	LANDSCAPE	
			PORTRAIT	
		PAGE-SIZE	A1	
			A2	
			A3	
			A4	
			A5	
			US-letter	

PURPOSE:

The command sets parameters for plotting. The settings must be done prior to giving the PLOT command.

PARAMETERS:

FILE	Set file prefix and name for the plot file. Default is the same as for the model file. The file extension depends on the type of format.
FORMAT	Set the plot format.
number	The plotter number may alternatively be given, but you will normally not know this.
SESAM-NEUTRAL	A plot format of the SESAM system. This is the default format. File extension is PLO.
POSTSCRIPT	The PostScript plot format. File extension is PS.
HPGL-7550	A Hewlett-Packard plot format. File extension is HP70.

HPGL-2	A Hewlett-Packard plot format. File extension is HPG2.
CGM-BINARY	The ISO 8632-3 Computer Graphics Metafile (CGM) plot format (binary encoding). File extension is CGM. This format is convenient for including plots in reports; see more information on this in Section 4.1.6.
COLOUR	Switch ON or OFF colours. The default is OFF. Colours are supported by the formats PostScript, HPGL-2 and CGM. Give this command after the SET PLOT FILE/FORMAT commands and prior to the PLOT command.
PAGE-SIZE	Set the plot page size. All sizes are not available for all plot formats. For SESAM-NEUTRAL this setting is irrelevant as the page size is set within the PLOT command. Give this command after the SET PLOT FILE/FORMAT commands and prior to the PLOT command.
A1 ... A5, US letter	Standard page sizes (paper formats). See explanation for the PLOT command. A4 is the default choice.
ORIENTATION	Set the page orientation.
LANDSCAPE	Landscape orientation. Only available for HPGL-2 and PostScript formats.
PORTRAIT	Portrait orientation. This is default.

NOTES:

For PostScript and HPGL-2 the size specification in the PLOT command is dummy. It will not change the plot size, the specification is retained for compatibility with old input files. For SESAM-NEUTRAL format the SET PLOT PAGE-SIZE has no effect as the size specification in the PLOT command is used.

SET PRINT

...	PRINT	DESTINATION	...
		FILE	
		FORMAT	
		PAGESIZE	
		TABLE	

PURPOSE:

The command sets different parameters controlling the execution of the PRINT command.

SET PRINT DESTINATION

...	DESTINATION	SCREEN
		FILE

PURPOSE:

The command switches the print between the screen and a file. The command PRINT ALL will regardless of this selection direct the print to a file.

PARAMETERS:

SCREEN Print will be directed to the screen.

FILE Print will be directed to a file.

SET PRINT FILE

...	FILE	LINEPRINTER	
		NAME	filnam

PURPOSE:

The command decides where the print to file is sent. It is either sent directly to the line printer or to the specified file. Print to file is, by default, sent to a file with the same name as the model (and command log) file. The file extension will always be LIS.

PARAMETERS:

LINEPRINTER	Connected line printer
NAME	Name of a print file is to be specified
filnam	File name given without the file extension

SET PRINT FORMAT

...	FORMAT	E
		F
		G

PURPOSE:

The command decides which FORTRAN format to use when tabulating certain data. By default F format is used whenever possible. Large real values cannot be printed in F format; the user may in such cases specify E or G format. Examples of output in each of the above formats are given below:

E formatted output

SUPER ELEMENT TYPE:		3		LEVEL:		1	
EXT.	INT.	I N I T I A L		C O N D T I O N			
NO.	NO.	RX	RY	RZ	TYPE		

101	1	0.5000000E+01	0.6000000E+01	0.7000000E+01	VELO.		
		0.8000000E+01	0.9000000E+01	0.1000000E+02			
105	2	0.1000000E+01	0.0000000E+00	0.8000000E+01	DISP.		
		0.5000000E+01	0.5000000E+01	0.0000000E+00			
105	2	0.1000000E+01	0.2000000E+01	0.3000000E+01	VELO.		
		0.4000000E+01	0.5000000E+01	0.6000000E+01			

F formatted output

SUPER ELEMENT TYPE:		3		LEVEL:		1	
EXT.	INT.	I N I T I A L		C O N D T I O N			
NO.	NO.	RX	RY	RZ		TYPE	

101	1	5.000000	6.000000	7.000000		VELO.	
		8.000000	9.000000	10.000000			
105	2	1.000000	0.000000	8.000000		DISP.	
		5.000000	5.000000	0.000000			
105	2	1.000000	2.000000	3.000000		VELO.	
		4.000000	5.000000	6.000000			

G formatted output

EXT. NO.	INT. NO.	I N I T I A L		C O N D T I O N	
		RX	RY	RZ	TYPE
101	1	5.000000	6.000000	7.000000	VELO.
		8.000000	9.000000	10.00000	
105	2	1.000000	0.0000000E+00	8.000000	DISP.
		5.000000	5.000000	0.0000000E+00	
105	2	1.000000	2.000000	3.000000	VELO.
		4.000000	5.000000	6.000000	

SET PRINT PAGESIZE

...	PAGESIZE	FILE	nlines
		SCREEN	

PURPOSE:

The command decides the number of lines printed for each page on the screen and on the file. The number of lines for each page on the screen is the number of lines printed between each time the user is asked whether to continue printing.

PARAMETERS:

FILE	Page size of a print file to be specified
SCREEN	Page size of the screen to be specified
nlines	Number of lines

SET PRINT TABLE NODE-BOUNDARY-TABLE

...	NODE-BOUNDARY-TABLE	DIGITS
		TEXT

PURPOSE:

The command switches between text or digit representation of the boundary condition codes. See the PRINT NODE BOUNDARY command.

PARAMETERS:

DIGITS	Digits will be used to represent the boundary condition codes in the print tables.
TEXT	Text description will be used to represent the boundary condition codes in the print tables.

SET SOIL-PROFILE-X-Y

...	SOIL-PROFILE-X-Y	x-value	y-value
-----	------------------	---------	---------

PURPOSE:

The command defines the global X and Y co-ordinates used for display of soil profile.

PARAMETERS:

x-value X co-ordinate to be used.

y-value X co-ordinate to be used.

NOTES:

The command should preferably be used prior to the ADD-DISPLAY SOIL-PROFILE command.

SET UNIT-VECTOR-TOLERANCE

...	UNIT-VECTOR-TOLERANCE	uvtol
-----	-----------------------	-------

PURPOSE:

The command specifies the unit vector tolerance used for deciding whether two vectors span a plane and whether a matrix is orthonormal. The unit vector tolerance has no unit and is initially set to 0.001.

PARAMETERS:

uvtol	Unit vector tolerance
-------	-----------------------

SET WRITE-MODE

...	WRITE-MODE	3DIMENSIONAL
		2DIMENSIONAL

PURPOSE:

The command is used to generate a 2 dimensional model as an alternative to the standard 3 dimensional model.

When generating the Input Interface File containing the FE model after giving the command SET WRITE-MODE 2DIMENSIONAL, the model is converted from a 3-D to a 2-D model by blocking the following three d.o.f.s: the translation in y and the rotations about x and z. Thus the 2-D model generated can be used directly for a 2-D analysis. Note that the program does not give any warning or message if the model generated contains data related to the d.o.f.s which are blocked during writing the 2-D model.

PARAMETERS:

3DIMENSIONAL	Three dimensional model
2DIMENSIONAL	Two dimensional model

SPLIT

SPLIT	ELEMENT-WISE	sub-commands
	NODE-WISE	

PURPOSE:

The command is used to split beam elements into two or many elements.

PARAMETERS:

ELEMENT-WISE Split by selecting element only.

NODE-WISE Split by selecting node and element.

The sub-commands and data are fully explained on the following pages.

SPLIT ELEMENT-WISE

...	ELEMENT-WISE	elno	ndiv	EVEN	...	nodeno*			...
				space*		STEP	startnode	stepno	
						AUTO			

...	elno*		
	STEP	startelem	stepele
	AUTO		

PURPOSE:

The command is used to split beam elements into two or many elements.

PARAMETERS:

elno	Beam element to split.
ndiv	Number of divisions. ndiv-1 nodes and ndiv-1 will be created.
EVEN	The element will be divided into ndiv equal part.
space	The spacings between the nodes starting from element start. If all ndiv spacings are entered they will be interpreted as relative spacings. If less than ndiv spacings are entered, they will be interpreted as true spacings and the remaining part will be divided into equal parts.
nodeno	Node number(s) of the node(s) to be generated.
STEP	Nodes/elements numbers will be generated step-wise.
startnode	Node number of first generated node.
stepno	The step in node numbering.
AUTO	Node/element numbers will be generated automatically. The numbers will be generated in sequence starting with the highest current node/element number plus one.
elno	Element number(s) of the element(s) to be generated.
startelem	Element number of first generated element.
stepele	The step in element numbering.

NOTES:

Generated elements inherits material and cross section properties from original beam element.

All loads given to a beam element prior to the split process will disappear.

SPLIT NODE-WISE

...	NODE-WISE	node	elno	dist	...	nodeno	...	elno
						AUTO		AUTO

PURPOSE:

The command is used to split a beam into two elements.

PARAMETERS:

node	Node to split from.
elno	Beam element to split.
dist	Distance (in length unit) from node to insert split node.
nodeno	Node number of the node to be generated.
AUTO	Node/element numbers will be generated automatically. The numbers will be generated in sequence starting with the highest current node/element number plus one.
elno	Element number of the element to be generated.

NOTES:

Generated elements inherits material and cross section properties from original beam element.

All loads given to a beam element prior to the split process will disappear.

TRANSFORMATION

TRANSFORMATION	trano	spx	spy	spz	gpx	gpy	gpz
----------------	-------	-----	-----	-----	-----	-----	-----

PURPOSE:

The command defines a transformed coordinate system. The transformation matrix transforms coordinates from a transformed coordinate system to the global coordinate system. A transformation can be used:

- For specifying a fixation or a prescribed displacement in a transformed coordinate system. See the BOUNDARY command.
- For specifying loads and eccentricities in a transformed coordinate system.
- For defining the orientation of general spring, shim, spring to ground, and damper to ground elements.
- For changing the coordinates of nodes. See the CHANGE NODE command.

The transformation matrix is defined by giving the global coordinates of a second point (SP) and a guiding point (GP). The x-axis of the transformed coordinate system, X_T , goes from the origin to SP. The transformed z-axis, Z_T , is perpendicular to X_T and so that GP lies in the X_T - Z_T plane on the positive Z_T side. Y_T is perpendicular to X_T and Z_T . See Figure 5.32.

PARAMETERS:

trano	Transformation reference number.
spx spy spz	Second point global coordinates.
gpx gpy gpz	Guiding point global coordinates.

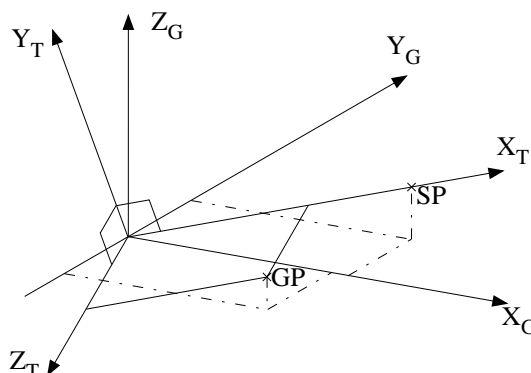


Figure 5.32 Definition of a transformed coordinate system

WRITE

WRITE	sup-el-no	
	BANDWIDTH-OPTIMIZATION	sup-el-no
	NO-OPTIMIZATION	sup-el-no
	PROFILE-OPTIMIZATION	sup-el-no
	END-CUT-DATA	
	GENSOD-SPLICE-TEMPLATE	...

PURPOSE:

The command writes an Input Interface File containing the FE model (the superelement). It is also used to write the tubular bracing end-cut data file to be used by the jacket launch program Installjac, and to write template input files used to run Gensod and Splice.

Several consistency checks of the model are performed during writing of the Input Interface File. For example:

- Are there nodes not connected to any elements?
- Are there elements not having any material assigned?
- Are there beam, truss or non-structural beam elements not having any section assigned?

In addition, default local coordinate systems are generated for elements for which local coordinate systems have not explicitly been defined. The orientation of the default local coordinate system is described for the PROPERTY LOCAL-COORDINATE command.

See Section 2.5 on the necessity of optimising the node numbering.

PARAMETERS:

sup-el-n	superelement number
BANDWIDTH-OPTIMIZATION	The node numbering is optimised with respect to the bandwidth of the stiffness matrix.
NO-OPTIMIZATION	The node numbering is not optimised.
PROFILE-OPTIMIZATION	The node numbering is optimised with respect to the profile of the stiffness matrix.
END-CUT-DATA	Write end-cut data to separate file for use in the Installjac launch program.
GENSOD-SPLICE-TEMPLATE	Writes the input files / templates to be used by Gensod (GENSOD.INP) and Splice (SPLICE.INP).

NOTES:

Giving the command WRITE sup-el-no involves no optimisation of the node numbering; see Section 2.5 on the consequence of this.

WRITE GENSOD-SPLICE-TEMPLATE

WRITE	GENSOD-SPLICE-TEMPLATE	soil-id	numvec	topsup
-------	------------------------	---------	--------	--------

PURPOSE:

The command writes the input files / templates to be used by Gensod (GENSOD.INP) and Splice (SPLICE.INP).

PARAMETERS:

soil-id	The soil profile id (number) to be used. Currently only id 1 allowed.
numvec	Number of load vectors to be analysed by Splice (default = number of basic load cases defined in Preframe).
topsup	Top level superelement number. Default = 21.

NOTES:

Always check the input file prior to running GENSOD / SPLICE. It may be necessary to manually modify the data, e.g. the numvec variable (due to seastate loads and load combinations) in the SPLICE input template.

Preframe is not able to read the top level superelement number given in Manager, hence this must be specified due to the Matrix Interface File reference on the SPLICE input template.

The gravity loading factor ACCZ on the SPLICE input template is set equal to the gz parameter in gravity load case number 1, with opposite sign. (If not defined, the value 1.0 is used.)

The value of Z-SCOUR on the SPLICE input template is set equal to the sum of ZSURF + SCRLOC from the GENSOD input template.

The WRITE command is not logged on the journal file.

ZOOM

ZOOM	IN	...
	OUT	

PURPOSE:

The command will in combination with manipulating the graphics cursor zoom in or out on the displayed picture.

How to use the mouse (left mouse button) to determine the zoom area may vary with the graphics device. On most devices, however, you may either:

- press and hold while dragging to the opposite corner of a rectangle and then release, or
- click first in one corner and secondly in the opposite corner.

The zoom area is the smallest square containing the rectangle.

ZOOM IN will magnify the part of the picture that is inside the zoom area.

ZOOM OUT will fit the picture into the zoom area.

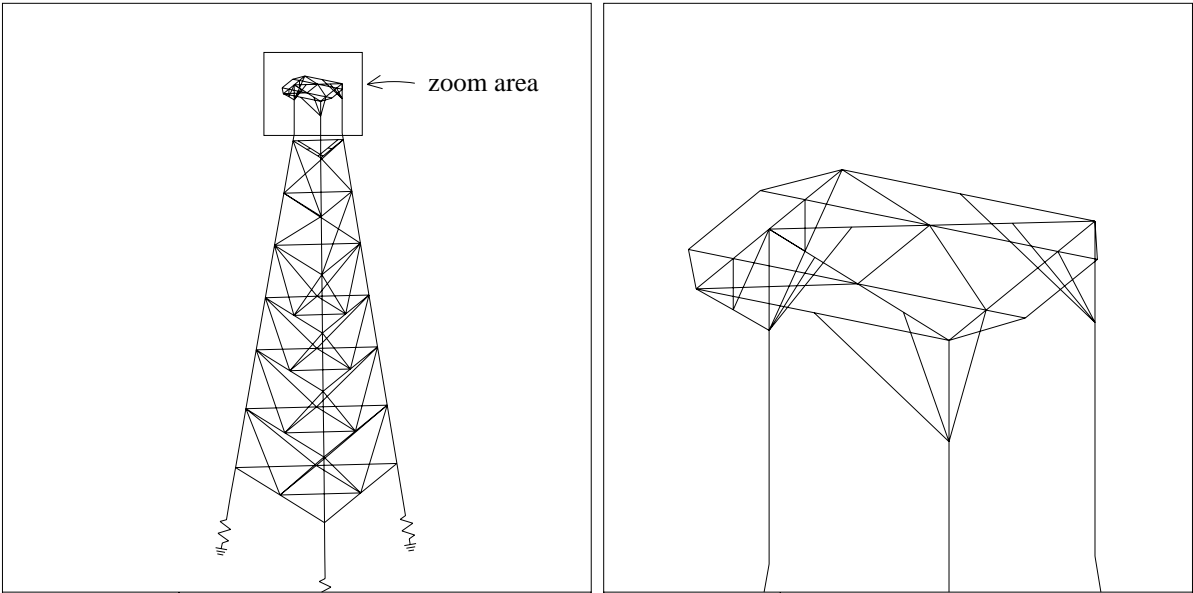


Figure 5.33 The effect of the ZOOM command

#

#	ncomnd
	ALL

PURPOSE:

The command reads commands from the command input file. The command input file is opened by the command SET COMMAND-INPUT-FILE. The command input file can either be a command log file from a previous run or a file prepared by a text editor.

The program will execute commands from the command input file until:

- an end-of-file is detected,
- a '#' is found on the file,
- ncomnd number of commands have been read, or
- an erroneous command sequence is found.

A 'command loop' is taken as a single command in this context. For example, '# 1' will read the command defining all three nodes from the file below, and '# 2' will read the whole file.

```

NODE 101 0. 0. 0.
      102 10. 0. 0.
      103 10. 10. 0.
. .
ELEMENT BEAM 11 101 102
              12 102 103
. .

```

PARAMETERS:

ncomnd Number of commands to be read from the command input file

ALL Read all commands from the command input file

NOTES:

See also note in connection with the command SET COMMAND-INPUT-FILE.

APPENDIX A TUTORIAL EXAMPLES

The following tutorial examples are presented:

- 1 Modelling a module frame, see Section A 1.
- 2 Modelling a small 4 legged jacket with soil and pile, see Section A 2.

Note that Section 3.2.4 provides details on how to model an 8 legged jacket with conductors using the GENERATE command.

Section A 3 illustrates the effect of the CHANGE JOINT sel-nodes GAP-PLANEWISE command.

A 1 Modelling a Module Frame

This example illustrates most basic modelling techniques of Preframe.

The model to create is illustrated in Figure A.1. The following consistent set of units are used: KiloNewton (kN), meter, second and tonne. The main dimensions are defined in Figure A.2. The cross sectional data are defined in Figure A.3. The material is steel with data: $E = 2.1E8$, $\nu = 0.3$ and $\rho = 7.85$. The loads are shown in Figure A.4. The module is fixed by spring-to-ground elements connected to the bottom of the four tubular support elements.

Hints for how to create the model using the graphical user interface as described in Section 3.1 are given below. You may therefore use this example as a tutorial in interactive use of Preframe. While the hints refer mostly to line-mode commands you may in this tutorial find it more convenient to use the equivalent graphical user interface actions.

The line-mode commands for creating the complete model are given at the end of the section. Note that the command log file from an interactive session will, more or less, be equal to the given line-mode commands.

If you don't bother giving a systematic node and element (beam) numbering you may give the command SET NUMBERING-AUTOMATIC ON. You will then not be prompted for node and element numbers. Note, however, that the log will contain the command SET NUMBERING-AUTOMATIC OFF and either AUTO or specific values for node and element numbers.

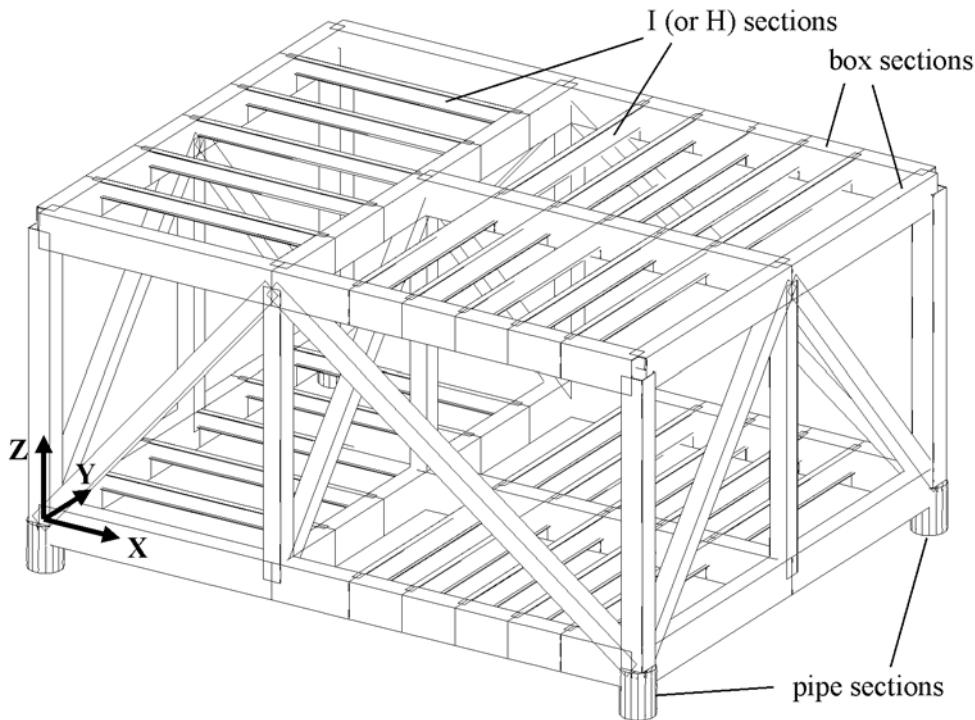


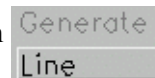
Figure A.1 A module frame

First model the cellar deck:

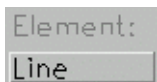
- Using the NODE command define the node at position cellar deck and intersection between axes A and 2 (i.e. a node in the origin) and the node at position cellar deck A/4 (28,0,0).
- Generate a line of nodes and elements between the two nodes. There should be 7 divisions (beam elements) along the line with a spacing as given in Figure A.2.

GENERATE BEAM LINE 'click 1st node' 'click 2nd node' 7 'give spacings'

Rather than entering GENERATE BEAM LINE you may click button



- Define a set containing all nodes and elements created so far:
DEFINE SET 'set-name' UNION-WITH NODE ALL UNION-WITH ELEMENT ALL END
- Copy the defined set a distance 11 in Y-direction (to create axis B in cellar deck):
COPY SET 'set-name' 0. 11. 0.
- Copy the defined set a distance 22 in Y-direction (to create axis C in cellar deck).
 - Note that this command will stop and give a warning: 'numbering failed, give new increment'. The reason for this is: Since we have requested automatic numbering of nodes and elements a default increment of 100 will be used. But this increment was used to create axis B so the numbers are taken.
 - Proceed by giving increments 200 for both nodes and elements.

- Click display button All (middle column of commands) to display the current model.
- Define all beam elements in Y-direction of cellar deck, i.e. along axes 2, 3 and 4 as well as in-between axes 3 and 4. Use the ELEMENT BEAM LINE command or button 
- Split beam elements along axes 2 and 3 in three equally long elements using the SPLIT ELEMENT-WISE command.
- Define beam elements in X-direction between axes 2 and 3 using the ELEMENT BEAM command.

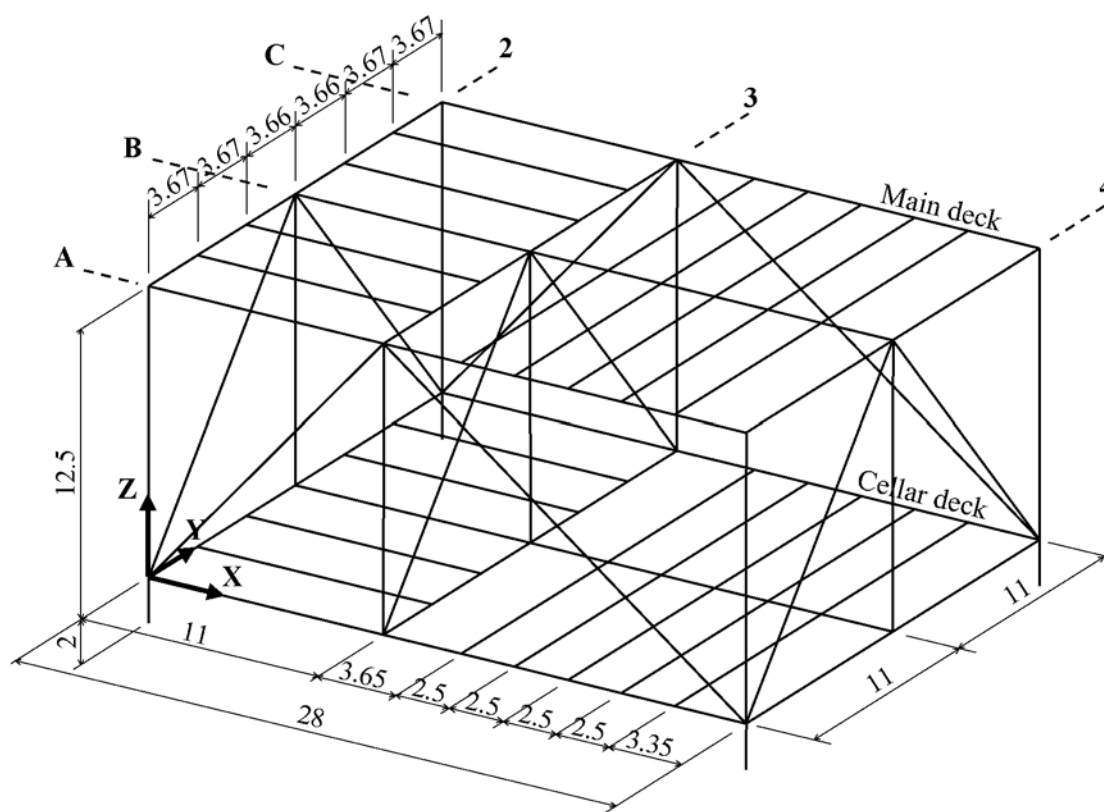
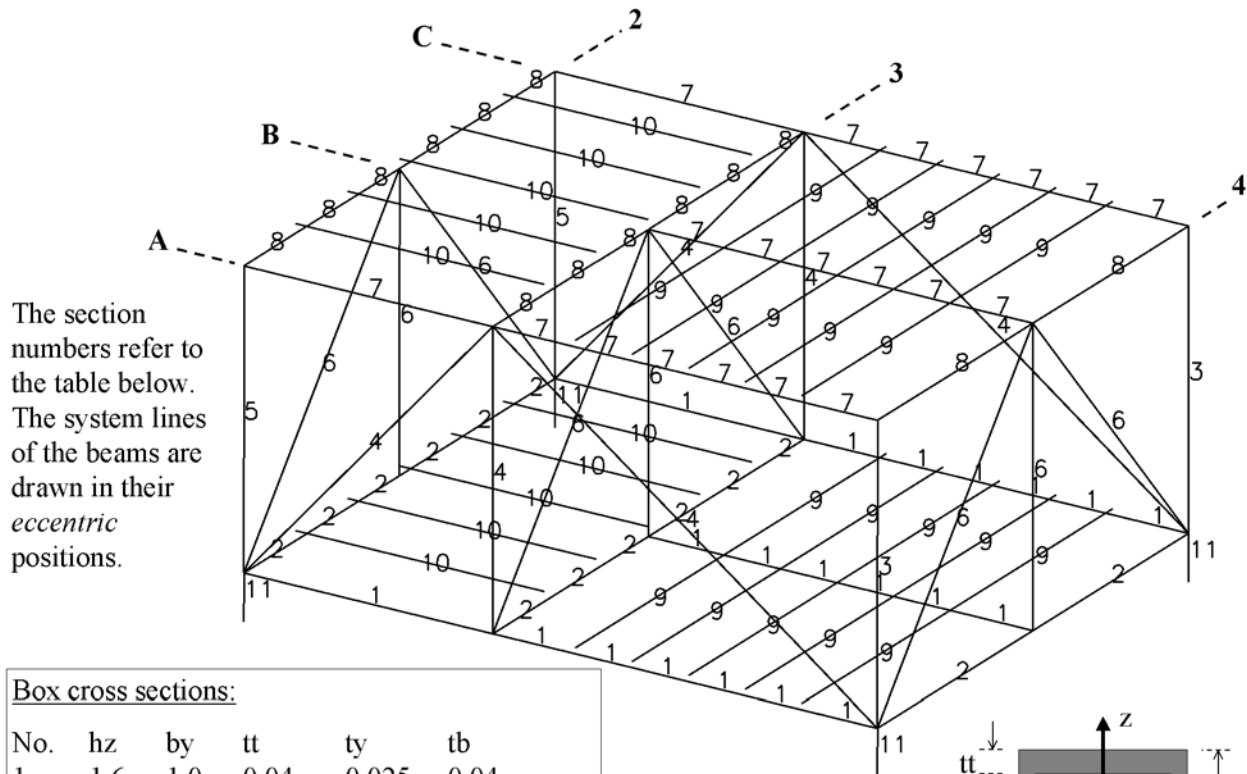


Figure A.2 The dimensions of the module frame

- Define the beam cross sections given in Figure A.3.
- Use the PROPERTY CONNECT SECTION command to assign section numbers to beam elements of the cellar deck. Refer to Figure A.3. Use the LINE option to select all elements between two nodes.
- Introduce eccentricities for all beams with section 9 (I-beam with height=1.0) so that their tops flush with beams with sections 1 and 2 (boxes with heights=1.6). The eccentricity will be half of the difference in section heights = $(1.6-1.0)/2 = 0.3$. Use the command
PROPERTY ECCENTRICITY BY-SECTION 9 NO GLOBAL 0.0 0.0 0.3 GLOBAL 0.0 0.0 0.3

- Introduce eccentricities for all beams with section 10 so that their tops flush with beams with sections 1 and 2. The eccentricity will in this case be $(1.6-0.7)/2 = 0.45$ in global Z-direction.
- Use the present button 'Facetted' (in middle column of commands) and zoom in to verify sections and eccentricities. Revert to wireframe display by clicking 'Wirefram'.



Box cross sections:

No.	hz	by	tt	ty	tb
1	1.6	1.0	0.04	0.025	0.04
2	1.6	0.8	0.04	0.025	0.04
3	1.0	1.0	0.05	0.05	0.05
4	1.0	0.8	0.035	0.035	0.035
5	1.0	1.0	0.035	0.035	0.035
6	0.8	0.8	0.025	0.025	0.025
7	1.5	1.0	0.04	0.025	0.04
8	1.5	0.8	0.04	0.025	0.04

I (or H) cross sections:

No.	hz	bt	tt	ty	bb	tb
9	1.0	0.3	0.036	0.019	0.3	0.036
10	0.7	0.3	0.032	0.017	0.3	0.032

Pipe cross section:

No.	dy	t
11	1.5	0.04

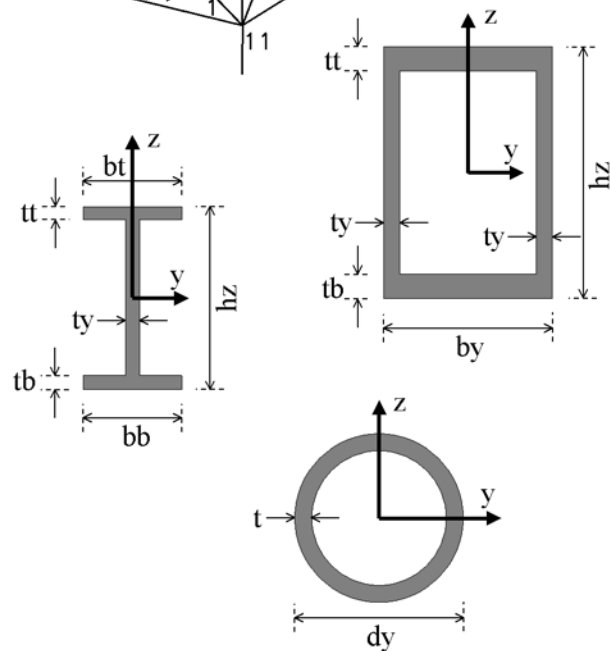


Figure A.3 The cross sections of the module frame

The cellar deck is complete. Now model the main deck.

- Define a set named LOWER containing all nodes and elements created so far.
- Copy this set to create the main deck (give node and element number increments 1000).
- Beams with box sections 1 and 2 in the cellar deck correspond to beams with box sections 7 and 8 in the main deck; see Figure A.3. Overrule previous section assignments with new ones by using the PROPERTY CONNECT SECTION command.
- The eccentricities of the I-beams in the main deck must be changed to flush with the new box sections. This may be done as follows: First define a set containing all elements in the main deck with section 9 and then refer to this set when changing the eccentricities:

```
DEFINE SET SECT-9 UNION-WITH ELEMENT BY-SECTION 9 NO SUBTRACT-BY ELEMENT  
SET LOWER NO END  
CHANGE ECCENTRICITY SET SECT-9 NO GLOBAL 0.0 0.0 0.25 GLOBAL 0.0 0.0 0.25
```
- Change eccentricities of I-beams in the main deck with section 10 in a similar way. The new eccentricity is 0.4.

The main deck is complete. Now model the braces and columns.

- Define elements for all columns and braces using the ELEMENT BEAM command.
- Connect sections to these elements.
- The default local coordinate systems assigned to the elements are OK for all elements except for the four braces between cellar and main deck along axes A and C; see Figure A.3. Use the command:

```
PROPERTY LOCAL-COORDINATE ZX-PLANE +Y-GLOBAL-INFINITY
```

to ensure that their local z-axes point in global Y-direction.

The braces and columns are complete. Now create the loads. See Figure A.4 for data.

- Load case 1 is gravity:

```
LOAD 1 GRAVITY YES 0.0 0.0 -9.81
```
- Load case 2 is a set of concentrated forces on beam elements. The command asks for the distance from node 1 of the beam elements to the load. Which end is node 1 depends on how the elements were defined. Find node 1 by using the command LABEL LOCAL-COORDINATE LOCAL-Z-AXIS which displays the element z-axes near node 1 of the elements. Assuming node 1 consistently has the lower Y-coordinate define load case 2 by:

```
LOAD 2 ELEMENT POINT 'click elements' GLOBAL 2000 0 -15000 END .6667 POINT 'click elements' GLOBAL 2000 0 -15000 END 8.0 END END
```
- Load case 3 is a set of concentrated forces in the same positions as for load case 2.
- Load case 4 is a set of element distributed loads. Since these are constant and distributed over the whole length of the elements the same value is given in both ends and with distance 0.0 from the ends.
- Use the ADD-DISPLAY LOAD command to verify the loads.

Create supports and material.

- Use the NODE RELATIVE command to create the four support nodes at elevation -2.
- Create the four support elements (ELEMENT BEAM). Then create the pipe section and connect it.
- Define a spring material by the command:

```
PROPERTY MATERIAL 1 SPRING-TO-GROUND STIFFNESS 6
```

```
60000.0  0.0  0.0  0.0  0.0  0.0
```

```
      60000.0  0.0  0.0  0.0  0.0
```

```
        800000.0  0.0  0.0  0.0
```

```
          1000000.0  0.0  0.0
```

```
            1000000.0  0.0
```

```
              1000000.0
```

- Define spring-to-ground elements (ELEMENT SPRING-TO-GROUND) at the bottom of the four support elements and refer to the spring material defined above.
- Define a steel material and connect it to all elements (this assignment will fail for the spring-to-ground elements which is OK).

Loads

- Load case 1 is gravity.
- Load case 2 is a set of forces on four members in main deck: two members along axis 3 and two members along axis 4 as shown to the right. The force components are: 2000, 0, -15000.
- Load case 3 is a set of forces acting in the same positions as load case 2. See illustration to the lower right. The force components are: -2000, 0, -15000.
- Load case 4 is element distributed loads on I-beams in main deck; see below. The components are 0, 0, -22, on beams in X-direction and 0, 0, -85, on beams in Y-direction.

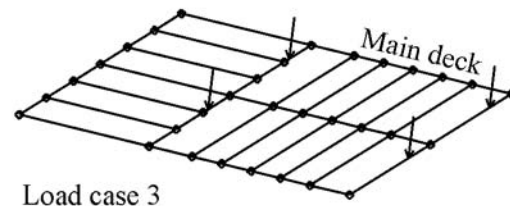
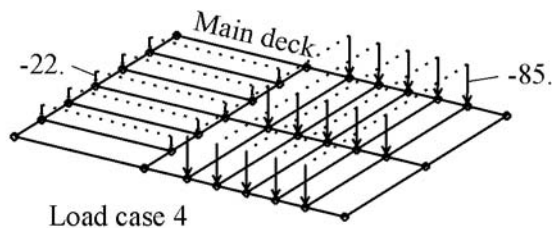
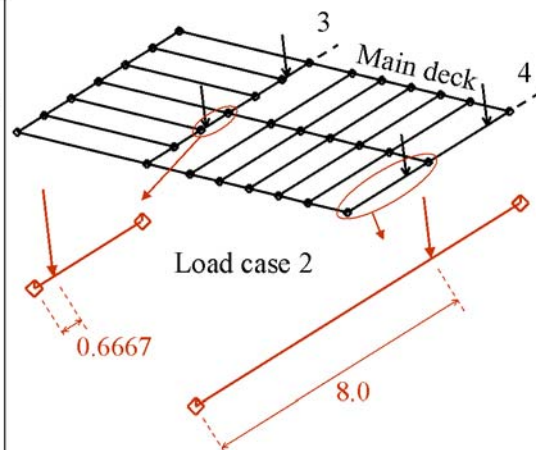


Figure A.4 The loads of the module frame

The line-mode commands for creating the complete model are given below.

```

NODE
1  0.0 0.0 0.0
2 28.0 0.0 0.0
..
%
% --- Generate nodes and elements along axis A in cellar deck
GENERATE BEAM LINE 1 2 7 AUTO AUTO
11.0 3.65 2.5 2.5 2.5 2.5 3.35
..
%
% --- Define a set containing nodes and elements and copy
DEFINE SET LINE_A UNION-WITH NODE ALL UNION-WITH ELEMENT ALL END
COPY SET LINE_A 0.0 11.0 0.0 100 100
COPY SET LINE_A 0.0 22.0 0.0 200 200
%
% --- Define beams of cellar deck in Y-direction
ELEMENT BEAM LINE
1 201 AUTO
3 203 AUTO
4 204 AUTO
5 205 AUTO
6 206 AUTO
7 207 AUTO
8 208 AUTO
2 202 AUTO
..
%
% --- Split beams along axes 2 and 3
SPLIT ELEMENT-WISE
8  3 EVEN AUTO AUTO
9  3 EVEN AUTO AUTO
10 3 EVEN AUTO AUTO
11 3 EVEN AUTO AUTO
..
%
% --- Define beams between axes 2 and 3
ELEMENT BEAM
32  9 13
33 10 14
34 11 15
35 12 16
..
%
% --- Define beam cross sections
PROPERTY SECTION
% --- box 1600 x 1000 x 25 x 40
1 BOX 1.6 1.0 0.04 0.025 0.04 1.0 1.0
% --- box 1600 x 800 x 25 x 40
2 BOX 1.6 0.8 0.04 0.025 0.04 1.0 1.0
% --- box 1000 x 1000 x 50 x 50
3 BOX 1.0 1.0 0.05 0.05 0.05 1.0 1.0

```

```
% --- box 1000 x 800 x 35 x 35
      4 BOX 1.0 0.8 0.035 0.035 0.035 1.0 1.0
% --- box 1000 x 1000 x 35 x 35
      5 BOX 1.0 1.0 0.035 0.035 0.035 1.0 1.0
% --- box 800 x 800 x 25 x 25
      6 BOX 0.8 0.8 0.025 0.025 0.025 1.0 1.0
% --- box 1500 x 1000 x 25 x 40
      7 BOX 1.5 1.0 0.04 0.025 0.04 1.0 1.0
% --- box 1500 x 800 x 25 x 40
      8 BOX 1.5 0.8 0.04 0.025 0.04 1.0 1.0
% --- HE1000B
      9 I 1.0 0.3 0.036 0.019 0.3 0.036 1.0 1.0
% --- HE700B
     10 I 0.7 0.3 0.032 0.017 0.3 0.032 1.0 1.0
..
%
% --- Connect sections to elements
PROPERTY CONNECT SECTION
1 LINE 1 2 LINE 103 102 LINE 201 202 NO
2 LINE 1 201 LINE 3 203 LINE 2 202 NO
9 12 13 14 15 16 17 18 19 20 21 NO
10 32 33 101 34 35 NO
..
%
% --- Give eccentricities to HE1000B beams (section 9)
PROPERTY ECCENTRICITY BY-SECTION 9 NO
GLOBAL 0.0 0.0 0.3 GLOBAL 0.0 0.0 0.3
%
% --- Give eccentricities to HE700B beams (section 10)
PROPERTY ECCENTRICITY BY-SECTION 10 NO
GLOBAL 0.0 0.0 0.45 GLOBAL 0.0 0.0 0.45
%
% --- Define a set containing all nodes and elements and copy to main deck
DEFINE SET LOWER UNION-WITH NODE ALL UNION-WITH ELEMENT ALL END
COPY SET LOWER 0.0 0.0 12.5 1000 1000
%
% --- Change some box sections of the main deck by connecting the proper ones
PROPERTY CONNECT SECTION
7 LINE 1001 1002 LINE 1103 1102 LINE 1201 1202 NO
8 LINE 1001 1201 LINE 1003 1203 LINE 1002 1202 NO
..
%
% --- Define set for HEB beams with section 9 in main deck only
DEFINE SET SECT-9 UNION-WITH ELEMENT BY-SECTION 9 NO
      SUBTRACT-BY ELEMENT SET LOWER NO END
%
% --- Define set for HEB beams with section 10 in main deck only
DEFINE SET SECT-10 UNION-WITH ELEMENT BY-SECTION 10 NO
      SUBTRACT-BY ELEMENT SET LOWER NO END
%
% --- Update eccentricities for HEB beams in main deck, refer to sets
CHANGE ECCENTRICITY SET SECT-9 NO
```

```

GLOBAL 0.0 0.0 .25 GLOBAL 0.0 0.0 0.25
CHANGE ECCENTRICITY SET SECT-10 NO
GLOBAL 0.0 0.0 .25 GLOBAL 0.0 0.0 0.4
%
% --- Define braces and columns
ELEMENT BEAM
36 1 1001
37 3 1003
38 2 1002
39 101 1101
40 103 1103
41 102 1102
42 201 1201
43 203 1203
44 202 1202
45 1 1003
46 2 1003
47 2 1102
48 202 1102
49 202 1203
50 201 1203
51 201 1101
52 1 1101
53 3 1103
54 203 1103
..
%
% --- Connect sections to braces and columns
PROPERTY CONNECT SECTION
3 38 44 NO
4 37 43 45 46 50 49 NO
5 36 42 NO
6 47 41 48 53 40 54 52 39 51 NO
..
%
% --- Change local z-axis for braces in axes A and B
PROPERTY LOCAL-COORDINATE ZX-PLANE +Y-GLOBAL-INFINITY 45 46 50 49 NO
..
%
% --- Define load case 1 (gravity)
LOAD 1 GRAVITY YES 0.0 0.0 -9.81
END
..
%
% --- Load case 2 is vertical + horizontal load in +X-direction
LOAD 2 ELEMENT
POINT 1029 1031 NO GLOBAL
2000.0 0.0 -15000.0 END 0.6667
POINT 1022 1023 NO GLOBAL
2000.0 0.0 -15000.0 END 8.0
END END END
%
```

```
% --- Load case 3 is vertical + horizontal load in -X-direction
LOAD 3 ELEMENT
POINT 1029 1031 NO GLOBAL
-2000.0 0.0 -15000.0 END 0.6667
POINT 1022 1023 NO GLOBAL
-2000.0 0.0 -15000.0 END 8.0
END END END
%
% --- Load case 4 is distributed load on main deck of 5 kN/m2.
% The load is applied as element distributed load on HE beams.
LOAD 4
ELEMENT DISTRIBUTED 1032 1033 1101 1034 1035 NO
GLOBAL 0.0 0.0 -22.0 END 0.0 0.0 0.0 -22.0 0.0
END
ELEMENT DISTRIBUTED 1012 1013 1014 1015 1016 1017 1018 1019 1020 1021 NO
GLOBAL 0.0 0.0 -85.0 END 0.0 0.0 0.0 -85.0 0.0
END END END
%
% --- Make supports at each corner
% a) Extra nodes
NODE RELATIVE
1 0.0 0.0 -2.0 4001
2 0.0 0.0 -2.0 4002
201 0.0 0.0 -2.0 4201
202 0.0 0.0 -2.0 4202
..
%
% --- b) Create 4 support elements
ELEMENT BEAM
4001 4001 1
4002 4002 2
4201 4201 201
4202 4202 202
..
%
% --- c) Define a pipe section and connect to elements
PROPERTY SECTION 11 PIPE 1.5 0.04 1.0 1.0
..
PROPERTY CONNECT SECTION 11 4001 4002 4201 4202 NO
..
%
% --- d) Define spring to ground stiffness (material)
PROPERTY MATERIAL 1 SPRING-TO-GROUND STIFFNESS 6
60000.0 0.0 0.0 0.0 0.0 0.0
60000.0 0.0 0.0 0.0 0.0 0.0
800000.0 0.0 0.0 0.0
1000000.0 0.0 0.0
1000000.0 0.0
1000000.0
END
%
% --- e) Define spring to ground elements
```

```

ELEMENT SPRING-TO-GROUND- (GSPR)
5001 4001 GLOBAL 1
5002 4002 GLOBAL 1
5201 4201 GLOBAL 1
5202 4202 GLOBAL 1
END
..
%
% --- Define material and connect it to all elements
PROPERTY MATERIAL 2 LINEAR-ELASTIC 2.1E+8 0.3 7.850 0.0 0.0
..
PROPERTY CONNECT MATERIAL 2 ALL
..

```

A 2 Modelling a Small 4 Legged Jacket with Soil and Piles

The modelling procedure is as follows:

- 1 Generate the jacket structure. (Global Z axis pointing upwards.)
- 2 Define the soil data, i.e.:
 - Mudline level
 - Soil types (sand / clay)
 - Soil profile (soil types and layer divisions)
 - Skin friction and tip resistance data
 - PY, TZ and QZ codes
- 3 Generate the piles:
 - Generate piles based on soil profile or user given segment lengths and number of elements
 - Add necessary pile data (attributes), i.e. yield strength, tip-code, ‘fixed-to’ node reference when pile groups, density of fluid inside piles
- 4 By use of the WRITE command, create the:
 - SESAM Input Interface File
 - The superelement containing the piles must be number 1, i.e. T1.FEM.**
 - Gensod input / template file
 - Splice input / template file

The Preframe commands for generating the model including soil data and piles are given in the following.

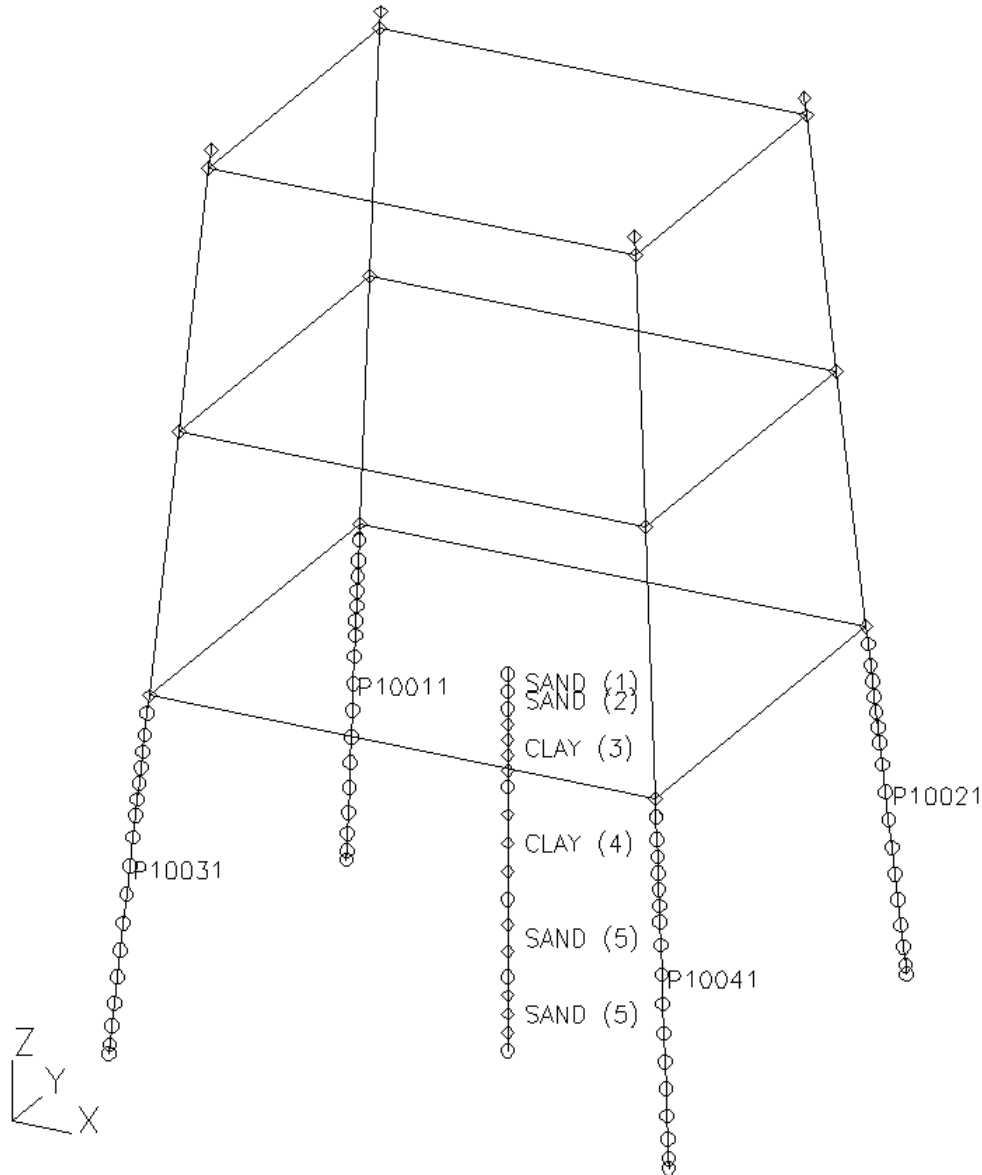


Figure A.5 4 legged jacket with piles

```
% UNITS: kN, m (g = 9.81 m/s2)
% -----
% 1. Generate the jacket structure
% -----
%
PROPERTY SECTION
1 PIPE 1.25 .06 1.0 1.0
2 PIPE .6 .025 1.0 1.0
3 PIPE .5 .02 1.0 1.0
% Section 4 is used for piles
```

```
4 PIPE 1.1 .05 1.0 1.0
END
%
PROPERTY MATERIAL 1 LINEAR-ELASTIC 210.E+06 0.3 7.85 0.0 0.12E-04
END
% Material 2 used for piles, density = 6.93 tonnes/m3
% equals "gamma-wall" = 68.0 kN/m3 on PILGEN.INP
PROPERTY MATERIAL 2 LINEAR-ELASTIC 210.E+06 0.3 6.93 0.0 0.12E-04
END
%
SET ALIGNMENT-AUTOMATIC ON
END
%
GENERATE BEAM-(BEAS) JACKET 4-LEGGED
60.0 50.0 -2.0 50.0 40.0 62.0
0.0 30.0 60.0 END
BRACINGS X-BRACINGS ALL-ROWS
1 2 END
SECTIONS LEGS ALL-ELEVATIONS 1
END
HORIZONTAL-BRACINGS ALL-ELEVATIONS 2
END
X-BRACINGS ALL-ELEVATIONS 3
END
END END
..
%
LOAD 1 GRAVITY YES 0.0 0.0 -9.81
END
..
LOAD 1 NODE FORCE 10411 10421 10431 10441 NO
GLOBAL 500.0 500.0 -70000.0 0.0 0.0 0.0 END
END END END
%
PROPERTY CONNECT MATERIAL 1 ALL
END
END
% -----
% 2. Define soil data and soil profile
% -----
PROPERTY SOIL
SAND 1 19.5 38.0 1.0 0 1.0 0.5E-02
SAND 2 19.5 32.0 1.0 0 1.0 0.5E-02
CLAY 3 19.0 100.0 100.0 0.1E-01 1.0 0.5 0 1.0 0.5E-02
CLAY 4 19.0 130.0 130.0 0.1E-01 1.0 0.5 0 1.0 0.5E-02
SAND 5 20.0 37.0 1.0 0 1.0 0.5E-02
END
%
DEFINE SOIL
PARAMETER MUDLINE-LEVEL -1.5 END
PROFILE-(LAYER) 1 6
-3.5 1 1
```

```

-5.5 1 2
-14.5 3 3
-27.5 4 4
-36.5 3 5
-45.0 3 5
END
%
ASSIGN SOIL-DATA
PY-TZ-QZ-CODE
Z-LEVEL -1.5 287 293 293
END
SKIN-FRICTION
Z-LEVEL -1.5 5.0 3.0 -1.0 0.1E-01 0.5 0.5E-01
Z-LEVEL -3.5 15.0 11.0 -1.0 0.1E-01 0.5 0.5E-01
Z-LEVEL -5.5 45.0 45.0 -1.0 0.1E-01 0.5 0.5E-01
Z-LEVEL -14.5 75.0 75.0 -1.0 0.1E-01 0.5 0.5E-01
Z-LEVEL -27.5 110.0 95.0 -1.0 0.1E-01 0.5 0.5E-01
END
TIP-RESISTANCE
Z-LEVEL -36.5 120.0 120.0 -1.0 0.1E-01 13350.0 0.5 0.5E-01
Z-LEVEL -101.5 120.0 120.0 -1.0 0.1E-01 14000.0 0.5 0.5E-01
END
..
% -----
% 3. Generate the piles
% -----
% Generate one pile in each corner and assign pile parameters / data
GENERATE BEAM-(BEAS) PILE-FROM-SOIL 1 MAIN BY-NODE-SELECT
10031 10041 10021 10011 NO
-40.75 4 2
..
%
ASSIGN PILE-DATA YIELD-STRENGTH 350.0E+03 CONCEPT ALL
TIP-CODE
%      0 = Pile tip is free
%      1 = Pile tip is fixed
%      2 = Pile infinitely long beneath tip
%      3 = 2 + modified axial stiffness
%      -ND = Pile infinitely long below node ND
1 ALL
%
% Density-fluid = 1.0 tonnes/m3 equals
% "gamma-fluid" = 9.81 kN/m3 on PILGEN.INP
DENS-FLUID 1.0 CONCEPT ALL
END
END
% -----
% 4. Write superelement and template input files to Gensod and Splice
% -----
WRITE BANDWIDTH 1 ;
WRITE GENSOD-SPLICE-TEMPLATE 1 1 21

```

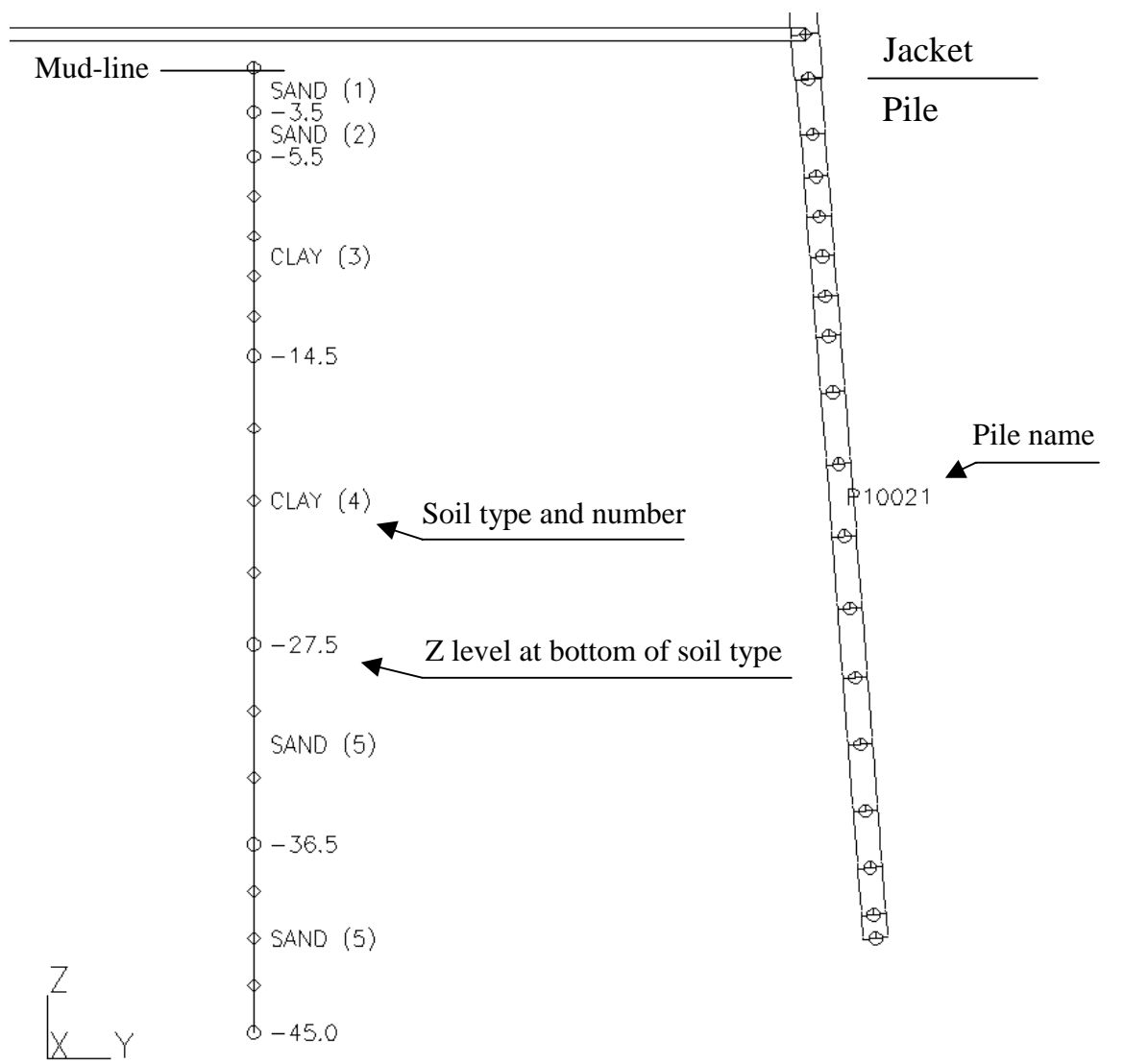



Figure A.6 Soil profile and piles, view in X direction

Example of Gensod and Splice templates (from above example):

GENSOD.INP:

```
GENSOD      - Project:
C
C   OPENED BY PREFRAME:  25-SEP-2001    09:00:34
C
C
C   ** W A R N I N G
C   ** This file is a template.
C   ** It MUST be checked before GENSOD is executed.
C   -----
```

```

***** CONTROL SECTION
1.000  CONFRC  OLD-FORCE-UNIT = CONFRC * NEW-FORCE-UNIT  1MN  = 1000*1KN
1.000  CONLTH  OLD-LENGTH-UNIT = CONLTH * NEW-LENGTH-UNIT  1M   = 3.28*1FT
5      NUMTYP  NUMBER OF DIFFERENT SOIL TYPES
13     NUMTQZ  NUMBER OF LINES IN THE T-Z / Q-Z DATA TABLE BELOW
15     NUMLAY  NUMBER OF SOIL LAYERS
0      NUMDSP  NUMBER OF Z-LEVELS WITH GIVEN SOIL DISPLACEMENTS
1      MIDBOT  P-Y ETC COMPUTED AT : 1=LAYER-MIDPOINT 2=LAYER-BOTTOM
          9.81 GAMMAW  UNIT WEIGHT OF WATER (9.81 KN/M3 IN SI-UNITS)
101.30 ATMPRS  ATMOSPHERIC PRESSURE (101.3 KN/M2 IN SI-UNITS)
          1.50 ZCYCL  Z-LEVEL DOWN TO WHICH CYCLIC P-Y DATA SHALL BE GENERATED
101.00 SUSTIF  USE STIFF CLAY P-Y PROCEDURES IF SU.GT.SUSTF (API ONLY)
2      JPRINT  PRINTED OUTPUT DATA (0=NONE 1=SOME 2=FULL)
1      JECHO   ECHO PRINT OF INPUT FILE NF14 TO FILE NF16 (0=NO 1=YES)

***** MATERIAL COEFFICIENTS SECTION
1.00   SFTPHI  MATERIAL COEFFICIENT ON TAN(PHI)
1.00   SFSU    MATERIAL COEFFICIENT ON UNDRAINED SHEAR STRENGTH
1.00   SFSKF   MATERIAL COEFFICIENT ON PILE SKIN FRICTION
1.00   SFSIGT  MATERIAL COEFFICIENT ON PILE TIP RESISTANCE

***** PILE DIAMETERS AND GROUP EFFECTS SECTION
1      NUMDIA  NUMBER OF PILE DIAM FOR WHICH P-Y/T-Z/Q-Z DATA IS WANTED
1.1
10000.00 ESOL0  E-SOIL FOR GROUP EFFECT CALCULATION :
1200.00 ESOL1   ESOL(Z) = ESOL0 + ESOL1*Z
0.50 POSAVR    SOIL AVERAGE POISSON RATIO FOR GROUP EFFECTS

***** SOIL SURFACE AND GROUND WATER SECTION
1.50 ZSURF    Z-LEVEL OF NON-SCoured SOIL SURFACE
2.00 SCRGEN   DEPTH OF GENERAL SCOUR BELOW ZSURF
4.00 SCRLOC   DEPTH OF LOCAL SCOUR BELOW ZSURF
20.00 SLOPE   SIDE SLOPE (DEGREES) OF LOCAL SCOUR HOLES
1.50 ZGRWT    Z-LEVEL OF GROUND WATER TABLE
9.81 GAMPWP   UNIT WEIGHT OF GROUND WATER (USED TO FIND PORE WATER PRSS)

***** LOADS AT SOIL SURFACE SECTION
0.00 SIGSRF   VERTICAL STRESS AT SURFACE
0.00 DPEMB    VERTICAL STRESS UNDER EMBANKMENT LOADING
0.00 AEMB     WIDTH A OF EMBANKMENT SLOPING PART
0.00 BEMB     PILE POSITION W.R.T. EMBANKMENT TOE (POSITIVE OUTSIDE)
0.00 DPCIRC   VERTICAL STRESS UNDER CIRCULAR LOADED AREA
0.00 RADIUS   RADIUS OF CIRCULAR LOADED AREA (PILE IS IN CENTER)
0      NUMFRC  NUMBER OF VERTICAL POINT FORCES AT SOIL SURFACE
0.00        POINT FORCE VALUES
0.00        HORIZONTAL DISTANCE TO PILE AXIS

***** SOIL TYPE AND PROPERTY SECTION
TYP  GAMTOT  PHI  SU.Z=0  SU.Z=100  EPSC  OCR  API-J  GAP  TR/TX  TZZR
1    0.1950E+02  38.00  0.0000E+00  0.0000E+00  0.00  1.00  0.00  0  1.00  0.005
2    0.1950E+02  32.00  0.0000E+00  0.0000E+00  0.00  1.00  0.00  0  1.00  0.005
3    0.1900E+02  0.00  0.1000E+03  0.1000E+03  0.01  1.00  0.50  0  1.00  0.005

```

```

4  0.1900E+02  0.00  0.1300E+03  0.1300E+03  0.01  1.00  0.50  0  1.00  0.005
5  0.2000E+02  37.00  0.0000E+00  0.0000E+00  0.00  1.00  0.00  0  1.00  0.005

```

***** SKIN FRICTION AND TIP RESISTANCE (T-Z/Q-Z) SECTION

10.0 ZONINF (ZONE OF INFLUENCE) / (PILE RADIUS) FOR TZCODE = 200

0.9 RFTZ CURVE FITTING FACTOR RF FOR TZCODE = 200

ZLEV	SKIN-CMP	SKIN-TNS	G0-SOIL	DSTZ/D	SIG-TIP	POIS	DSQZ/D
1.49	0.0000E+00	0.0000E+00	-0.1000E+01	0.00	0.0000E+00	0.00	0.00
1.50	0.5000E+01	0.3000E+01	-0.1000E+01	0.01	0.0000E+00	0.50	0.05
3.49	0.5000E+01	0.3000E+01	-0.1000E+01	0.01	0.0000E+00	0.50	0.05
3.50	0.1500E+02	0.1100E+02	-0.1000E+01	0.01	0.0000E+00	0.50	0.05
5.49	0.1500E+02	0.1100E+02	-0.1000E+01	0.01	0.0000E+00	0.50	0.05
5.50	0.4500E+02	0.4500E+02	-0.1000E+01	0.01	0.0000E+00	0.50	0.05
14.49	0.4500E+02	0.4500E+02	-0.1000E+01	0.01	0.0000E+00	0.50	0.05
14.50	0.7500E+02	0.7500E+02	-0.1000E+01	0.01	0.0000E+00	0.50	0.05
27.49	0.7500E+02	0.7500E+02	-0.1000E+01	0.01	0.0000E+00	0.50	0.05
27.50	0.1100E+03	0.9500E+02	-0.1000E+01	0.01	0.0000E+00	0.50	0.05
36.49	0.1100E+03	0.9500E+02	-0.1000E+01	0.01	0.0000E+00	0.50	0.05
36.50	0.1200E+03	0.1200E+03	-0.1000E+01	0.01	0.1335E+05	0.50	0.05
101.50	0.1200E+03	0.1200E+03	-0.1000E+01	0.01	0.1400E+05	0.50	0.05

***** SOIL LAYER DIVISIONS AND CODES FOR PY/TZ/QZ SECTION

P-Y CODES : 000=Manual 100+N=Auto 284=API-84 287=API-87 380=DNV-80

T-Z CODES : 000=Manual 100+N=Auto 200=Kraft et al. 293=API-93

Q-Z CODES : 000=Manual 100+N=Auto 200=bi-linear 293=API-93

LAYERS	TYP	ZBOTM	PYCODE	TZCODE	QZCODE
1	1	1	3.50	287	293
2	2	2	5.50	287	293
3	5	3	14.50	287	293
6	9	4	27.50	287	293
10	12	5	36.50	287	293
13	15	5	45.00	287	293

***** MANUAL P-Y (LATERAL RESISTANCE) DATA SECTION

LAY DIAM NUMPNT LINE1 : P-VALUES (F/L**2) LINE2 : Y-VALUES (L)

***** P-Y (LATERAL RESISTANCE) DATA MODIFICATION SECTION

LAYERS	DIAM	P-FACT	Y-FACT	Y-GAP	P-PEAK-FACT	P-RESID-FACT	Y-RESID-FACT
0	0	0.0	1.0	1.0	0.0	1.00	1.0

***** MANUAL T-Z (SKIN FRICTION) DATA SECTION

LAY DIAM NUMPNT LINE1 : T-VALUES (F/L**2) LINE2 : Z-VALUES (L)

***** T-Z (SKIN FRICTION) DATA MODIFICATION SECTION

LAYERS	DIAM	CMP-FCT	TNS-FCT	TZZ-FCT	T-RESID-FCT	Z-RESID-FCT
0	0	0.0	1.0	1.0	1.0	1.0

***** MANUAL Q-Z (PILE TIP RESISTANCE) DATA SECTION

LAY DIAM NUMPNT LINE1 : Q-VALUES (F/L**2) LINE2 : Z-VALUES (L)

***** Q-Z (PILE TIP RESISTANCE) DATA MODIFICATION SECTION

LAYERS	DIAM	CMP-FCT	TNS-FCT	QZZ-FCT	Q-RESID-FCT	Z-RESID-FCT
--------	------	---------	---------	---------	-------------	-------------

```
0 0      0.0      1.0      0.0      1.0      1.0      1.0
```

```
***** GIVEN SOIL DISPLACEMENTS AND OPEN HOLE SECTION
ZLEV  DSP-X  DSP-Y  DSP-Z  HOLE-DIAM
```

END OF GENSOD INPUT FILE

SPLICE.INP:

SPLICE - Project:

C

C OPENED BY PREFRAME: 25-SEP-2001 09:00:34

C

C

C ** W A R N I N G

C ** This file is a template.

C ** It MUST be checked before SPLICE is executed.

C

```
***** CONTROL SECTION
```

```
1.000  CONFRC  OLD-FORCE-UNIT = CONFRC * NEW-FORCE-UNIT 1MN = 1000*1KN
```

```
1.000  CONLTH  OLD-LENGTH-UNIT = CONLTH * NEW-LENGTH-UNIT 1M = 3.28*1FT
```

```
4      NPH     NUMBER OF PILE HEADS INCLUDING DUMMIES
```

```
4      NLPH    NUMBER OF PILE HEADS WITH GIVEN LOADS
```

```
1      JACK    CODE FOR PRESENCE OF JACKET (0=NO 1=YES)
```

```
0      LOAPIL  CODE FOR PRESENCE OF LOADS FROM PILGEN (0=NO 1=YES)
```

```
15     NUMLAY  NUMBER OF SOIL LAYERS
```

```
0      NSDSP   NUMBER OF Z-LEVELS WITH GIVEN SOIL DISPLACEMENTS
```

```
1      NUMVEC  NUMBER OF LOAD VECTORS TO BE ANALYZED
```

```
00     ISTART  CODE IJ FOR SAVE/READ OF RE-START VALUES (I=SAVE J=READ)
```

```
0      ISECM   CODE FOR SECOND ORDER MOMENTS (0=NEGLECT 1=INCLUDE)
```

```
0111   IPRT1   PRINT CODE IJKL (L=PILES K=LOADS J=SOIL I=SOIL.DISP)
```

```
0      JECHO   ECHO PRINT OF INPUT FILE NF5 TO FILE NF14 (0=NO 1=YES)
```

```
0 0 0 0 0     MISC(1) - MISC(5) SPECIAL PURPOSE PARAMETERS
```

```
***** NAME AND FORMAT OF MATRIX INTERFACE FILE WITH
```

C SUPERSTRUCTURE CONNECTION STIFFNESS MATRIX AND LOAD.

C PREFIX (E.G. DIRECTORY NAME) OF MATR. INTERF. FILE NAME.

C DEFAULT IS BLANK

C NAME FILE NAME OF MATRIX INTERFACE FILE.

M21

C FORMAT FORMAT OF MATR. INTF. FILE (FORMATTED/UNFORMATTED/NORSAM).

C DEFAULT IS UNFORMATTED (spec. for a file of type .SIU)

FORMATTED

```
***** POSTITION OF SPLICE/PILE GROUP COORD. SYST IN
```

C SUPERSTRUCTURE COORDINTATE SYSTEM

```
180.000 ALPHA  ANGLE TO ROTATE THE PILEGROUP AROUND THE SUPERSTRUCTURE
```

```
0.000 BETA     GLOBAL X, RESP. Y-AXIS TO HAVE PILEGROUP Z-AXIS DOWN.
```

```
0.000 DELTAX   THE X, Y AND Z-COORDINATES IN THE SUPERSTRUCTURE COOR-
```

```

    0.000 DELTAY      DINATE SYSTEM OF THE ORIGIN OF THE PILEGROUP.
    0.000 DELTAZ
C COS11...COS33; ALTERNATIVE TO ALPHA/BETA. DIRECTION COSINES OF THE COORDINATE-
C SYSTEM OF THE PILEGROUP IN THE SUPERSTRUCTURE COORDINATE
C SYSTEM. FIRST COLUMN IS THE DIR. COSINES OF THE X AXIS A.S.O.
C COS11      COS12      COS13
    0.0        0.0        0.0
C COS21      COS22      COS23
    0.0        0.0        0.0
C COS31      COS32      COS33
    0.0        0.0        0.0

***** PILE/SOIL/PILE INTERACTION AND PILE SCOUR SECTION
    0      INTER      MINDLIN INTERACTION CODE (0=NO 1=DIF.PILES 2=FULL -1=AVAIL)
    0.000 DISTIN      MAXIMUM INTERACTION DISTANCE
PILE1  PILE2  Z-SCOUR  (VALUES ARE READ UNTIL PILE2=NPH)
    1      4      5.50

***** POINT FORCES SECTION
    0      IFRC      NUMBER OF GIVEN POINT FORCES
FORCE   X-COORD  Y-COORD  Z-COORD      X-FORCE  Y-FORCE  Z-FORCE

***** LOAD APPLICATION SECTION
0.0 0.0  9.81      ACCX,ACCY,ACCZ  GRAVITY LOADING IN X- Y- AND Z-DIR
VECTOR  NUMINC  KEEP  VECMLT  SOLMLT  FRCMLT
    1      1      0      1.0      1.0      1.0
VECTOR  INCR  MAXIT  IPRT2  IPRT3  NPTRC  QTOT  CONV      MISC(6,7,8,9,10,11)
    1      1      20      0011  0111      1      1.0  .001      1 0 0 0 0 0
C
C VECTOR : LOAD VECTOR NUMBER 1,2,3,...,NUMVEC
C NUMINC : NUMBER OF LOAD INCREMENTS FOR EACH VECTOR
C KEEP : START CONDITION (0=ZERO 1=RESULTING CONDITION AFTER LAST VECTOR)
C VECMLT : LOAD VECTOR MULTIPLIER FOR LOAD VALUES READ FROM FILE NF9
C SOLMLT : SOIL DISPL MULTIPLIER FOR VALUES READ FROM FILES NF10 AND NF7
C FRCMLT : POINT FORCE MULTIPLIER FOR POINT FORCES GIVEN ABOVE
C
C VECTOR : LOAD VECTOR NUMBER 1,2,3,...,NUMVEC
C INCR : LOAD INCREMENT NUMBER FOR PRESENT VECTOR 1,2,3,...,NUMINC
C MAXIT : MAXIMUM NUMBER OF ITERATIONS
C IPRT2 : PRINT CODE IJKL (L=RES.DSP K=RES.FRC J=INC.DSP I=INC.FRC)
C IPRT3 : PRINT CODE IJKL (L=MATRIX K=PIL.RES J=SUMMARY I=NF15.SAVING)
C K=2 GIVES LIMITED PILE OUTPUT WITH DEPTH
C I=1 SAVES INTERFACE DISPL AND FORCES
C I=2 SAVES ALL DATA IN COMMON BLOCK /PILPRP/ , I=3 DOES BOTH
C NPTRC : TRACE PRINT OF PILE NPTRC DURING ITERATIONS
C QTOT : TOTAL LOAD (DISP) AFTER PRESENT INCR (INCR=VECT*(QTOTNOW-QTOTLAST))
C CONV : CONVERGENCE CRITERION (LENGTH)
C
C MISC(1) - MISC(11) ARE SPECIAL PURPOSE PARAMETERS
C
C MISC(1) : 1=CHECK EQUATION SOLVER 2=PRINT VECTOR AND SUB-MATRICES
C MISC(2) : 1=SKIP STIFFNESS MATRIX SYMMETRY ENFORCEMENT AT PILE HEADS

```

```

C MISC(3) : CONVERGENCE FACTOR DISPL ( NEXT=(OLD*F+NEW)/(1+F) F=MISC3/100 )
C MISC(4) : CONVERGENCE FACTOR FORCS ( NEXT=(OLD*F+NEW)/(1+F) F=MISC4/100 )
C MISC(5) : CONVERGENCE FACTOR STFNS ( NEXT=(OLD*F+NEW)/(1+F) F=MISC5/100 )
C MISC(6) = 0 COMPUTE NEW SOIL DISPL DUE TO GROUP EFFECTS AFTER EACH ITERATION
C           = 1 COMPUTE SOIL DISPL ONCE ONLY (SHOULD SECURE CONVERGENCE)
C MISC(7) = 1 PRINT INCR DSP AND FRC TO SCREEN AFTER EACH ITERATION
C MISC(8) = 1 PRINT STIFFNESS VALUES TO SCREEN AFTER EACH ITERATION
C MISC(9) = 9 PRINT PROGRAM CONTROL FLOW TO SCREEN
C MISC(9) = 8 SKIP AUTOMATIC DIVERGENCE CHECK
C MISC(10)= NPND TRACE PRINT OF VALUES FOR NODE ND ON PILE NP TO SCREEN
C MISC(11)= NF DETAILED ELEMENT DATA TRACE SAVED ON UNIT NF
END OF SPLICE INPUT FILE

```

A 3 Result of CHANGE JOINT sel-nodes GAP-PLANEWISE

The following pages shows some examples of the resulting element eccentricities after use of the CHANGE JOINT sel-nodes GAP-PLANEWISE command.

YT-joint:

The brace closest to the perpendicular to the chord will not be moved, i.e. eccentricities will be applied to the Y-brace to obtain the given gap value.

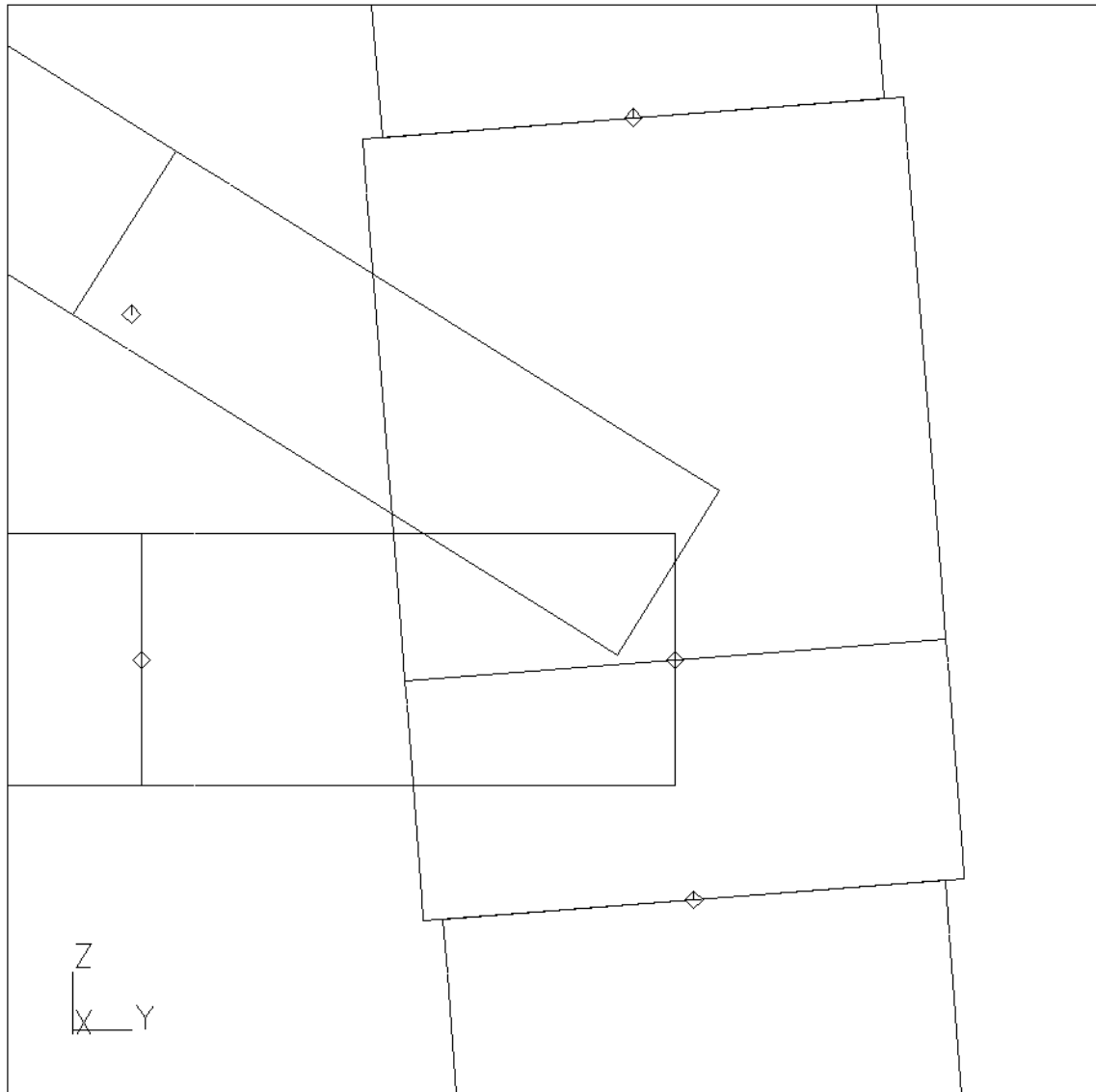


Figure A.7 YT-joint

KT-joint:

The brace in the middle will not be moved, i.e. eccentricities will be applied to the K-braces to obtain the given gap value.

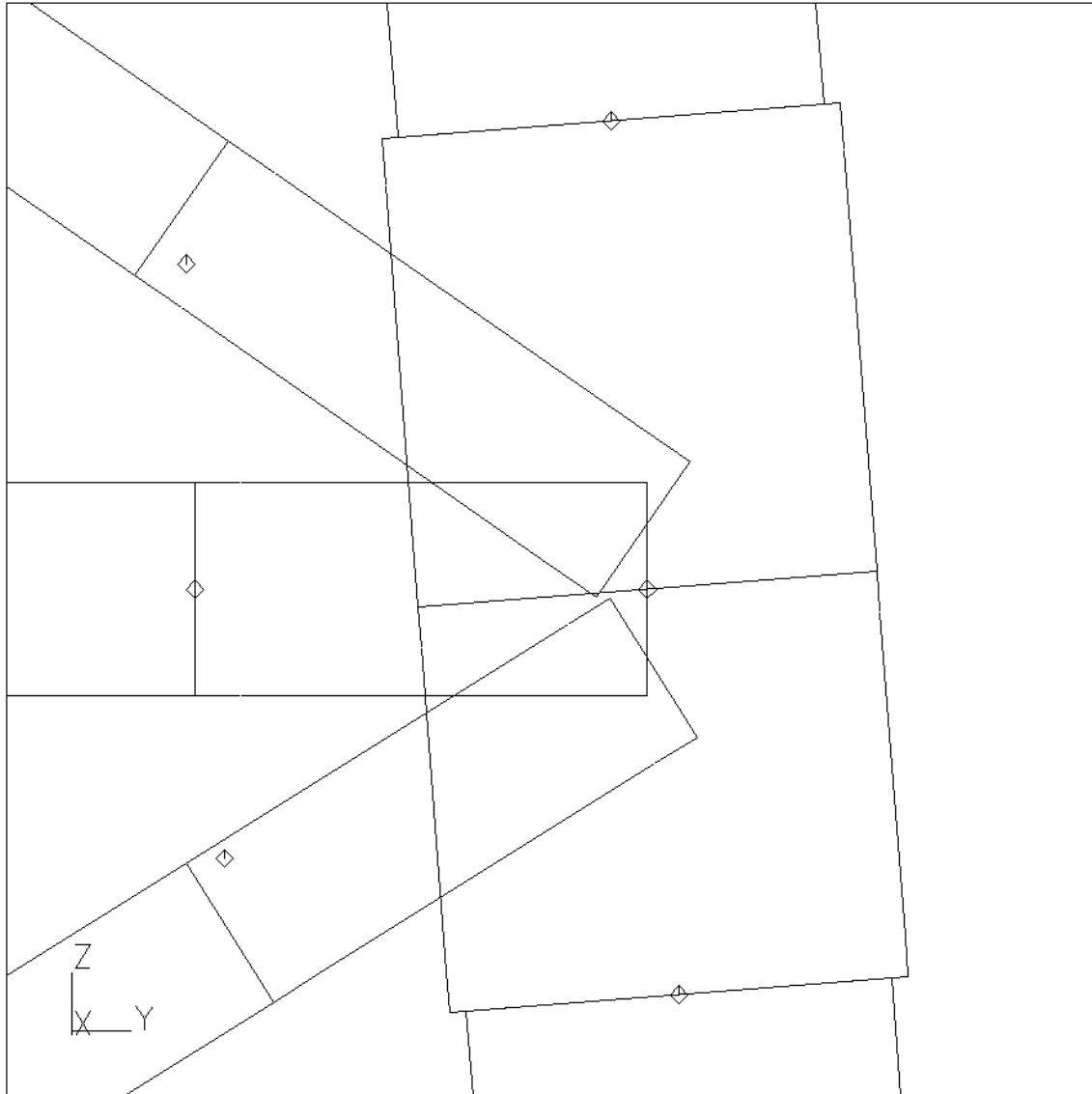


Figure A.8 KT-joint

Symmetric K-joint:

For symmetric K-joints, each of the two braces will be moved equally, i.e. half of the given gap value. The braces are defined as symmetric when the difference between the angles between the braces and the perpendicular to the chord is less than 5 degrees (inclusive estimated eccentricity).

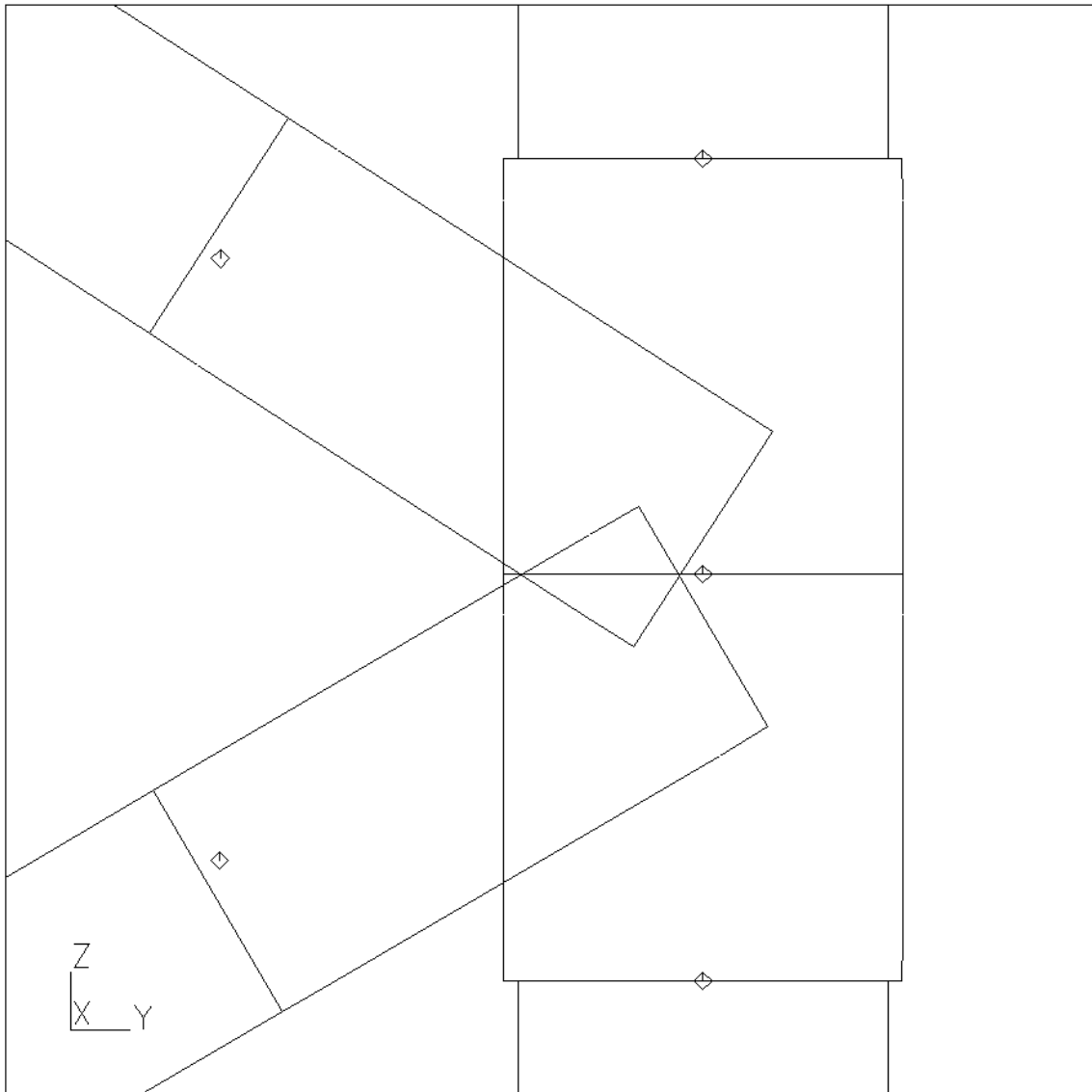


Figure A.9 Symmetric K-joint

Non-symmetric K-joint:

For non-symmetric K-joints only the brace with the smallest inclination to the chord will be moved, i.e. eccentricities will be applied to obtain the given gap value.

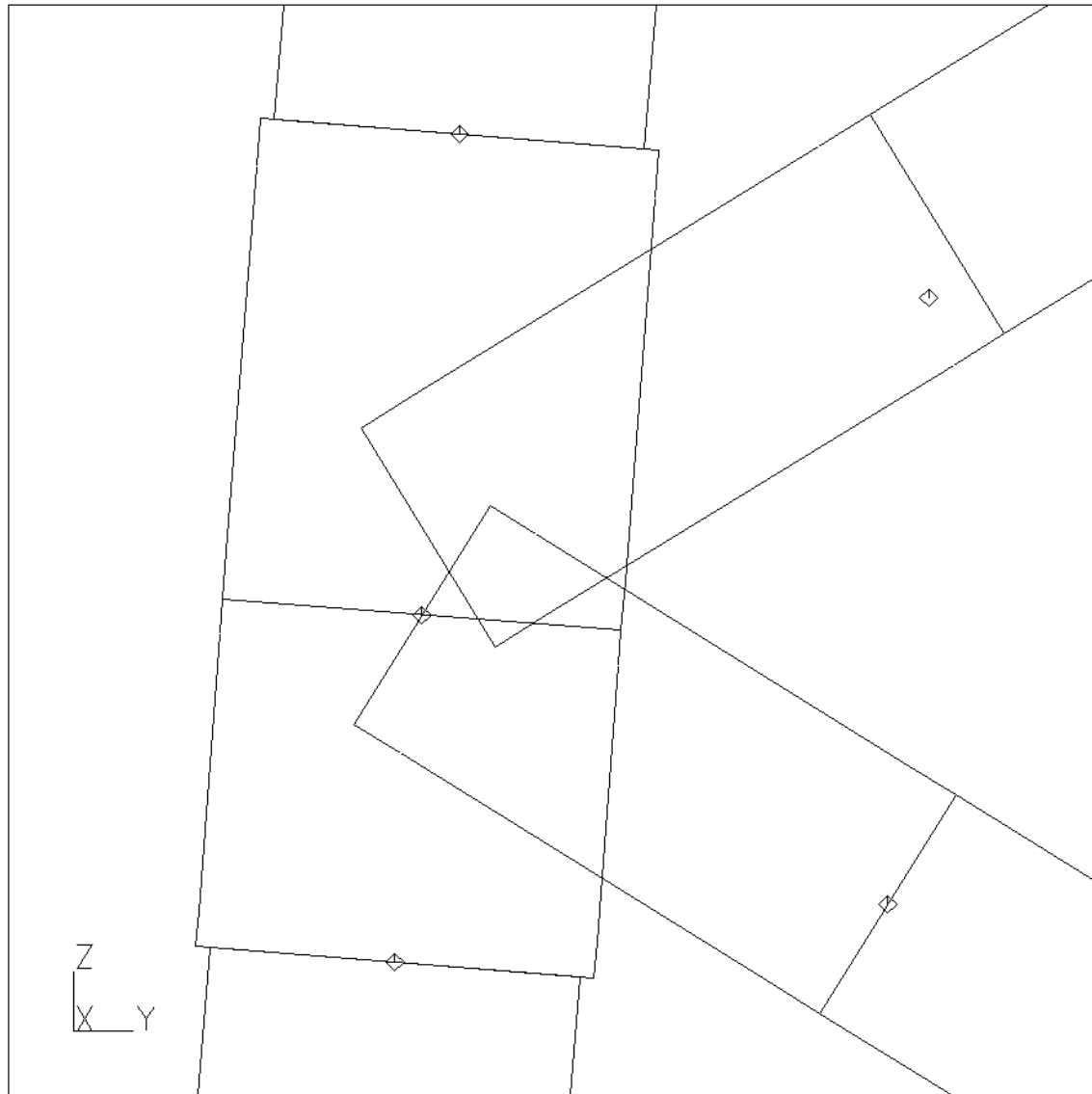


Figure A.10 Non-symmetric K-joint

APPENDIX B THEORY

B 1 Formulae for Sectional Parameters

Note: This section is incomplete in that the formulae for sectional parameters are lacking. The formulae may be found in the Prefem User Manual.

The formulae employed in Preframe for computing the sectional parameters for the various beam cross sections are given in the following. The formulae are taken from Ref. /1/, Ref. /2/ and Ref. /3/.

The following notation is used:

<i>AREA</i>	Cross sectional area
<i>IX</i>	Torsional moment of inertia about shear centre
<i>IY</i>	Moment of inertia about y-axis
<i>IZ</i>	Moment of inertia about z-axis
<i>IYZ</i>	Product of inertia about y- and z-axes
<i>WXMIN</i>	Minimum torsional sectional modulus about shear centre
<i>WYMIN</i>	Minimum sectional modulus about y-axis
<i>WZMIN</i>	Minimum sectional modulus about z-axis
<i>SHARY</i>	Shear area in the direction of y-axis
<i>SHARZ</i>	Shear area in the direction of z-axis
<i>SHCENY</i>	Shear centre location y-component
<i>SHCENZ</i>	Shear centre location z-component

SY	Static area moment about y-axis
SZ	Static area moment about z-axis
CY	Centroid location from bottom right corner y-component
CZ	Centroid location from bottom right corner z-component

Variables other than the ones above are only temporary.

Note: The local x-axis of the beam or truss element goes through the centroid of the cross section. I.e. the nodal displacements and consequently the cross sectional constants above refer to this axis. The torsional moment of inertia, however, refers to the shear centre. In most beam element theories the torsional d.o.f. is not coupled to the transverse d.o.f.s. Therefore, when torsion is of importance the shear centre should not be located far away from the centroid of the cross section, i.e. avoid heavily un-symmetrical cross sections.

B 1.1 Bar section

B 1.1.1 Sectional Dimensions

HZ	Height
BB	Width at bottom
BT	Width at top
SFY	Shear factor y-direction
SFZ	Shear factor z-direction

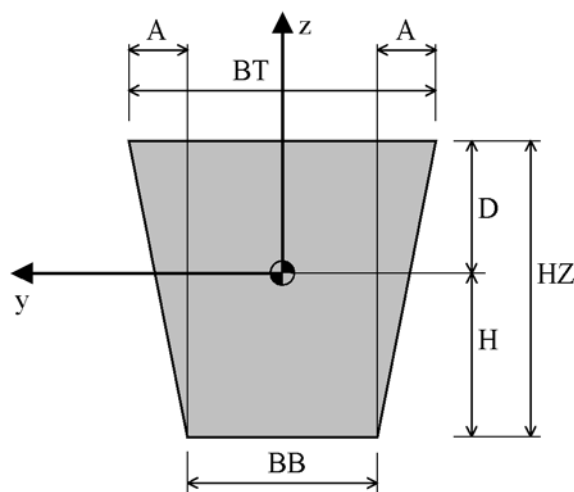


Figure B.1 Bar section

B 1.1.2 Sectional Parameters Computed

The expressions below for I_X , I_Y , I_Z , W_{XMIN} , W_{YMIN} , W_{ZMIN} , $SHARY$ and $SHARZ$ are taken from Ref. /1/. The expressions for $SHCENY$ and $SHCENZ$ are taken from Ref. /3/.

B 1.2 Box section

B 1.2.1 Sectional Dimensions

HZ	Height
BY	Width
TT	Thickness of top flange
TY	Thickness of webs (vertical walls)
TB	Thickness of bottom flange
SFY	Shear factor y-direction
SFZ	Shear factor z-direction

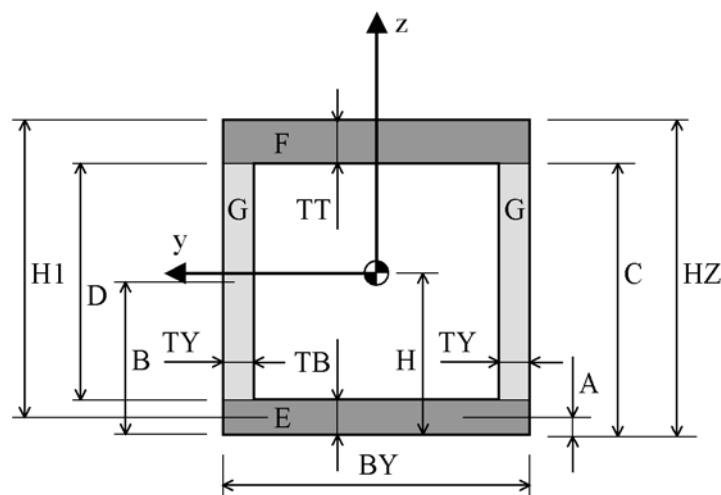


Figure B.2 Box section

B 1.2.2 Sectional Parameters Computed

The expressions below for I_X , I_Y , I_Z , W_{XMIN} , W_{YMIN} , W_{ZMIN} , $SHARY$, $SHARZ$ and $SHCENY$ are taken from Ref. /1/. The expression for $SHCENZ$ is taken from Ref. /2/.

B 1.3 Channel section

B 1.3.1 Sectional Dimensions

HZ	Height
BY	Width of top and bottom flanges
TZ	Thickness of top and bottom flanges
TY	Thickness of web
SFY	Shear factor y-direction
SFZ	Shear factor z-direction
$POSWEB$	=1 for web location in positive y-direction, otherwise =-1

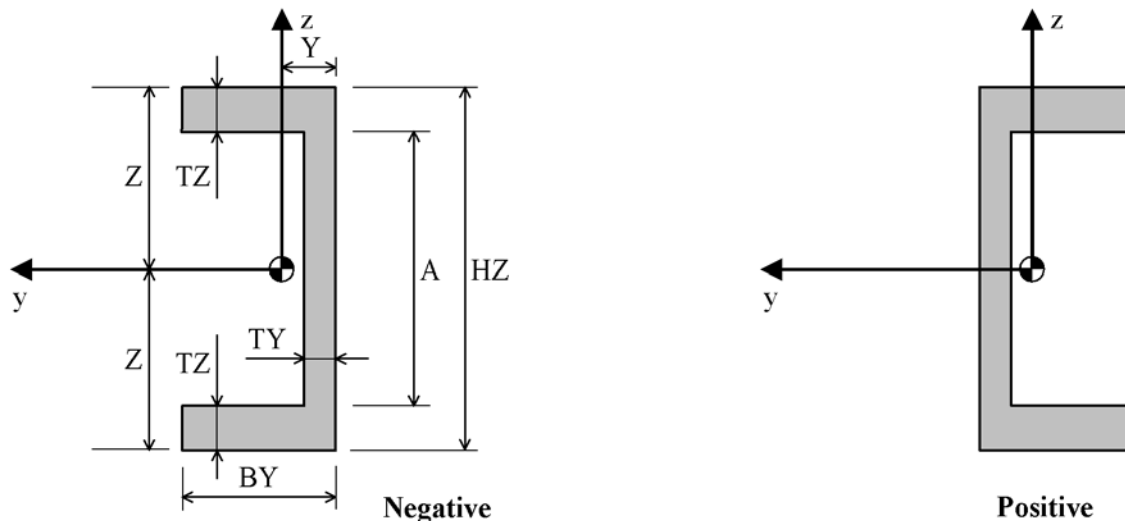


Figure B.3 Channel section

B 1.3.2 Sectional Parameters Computed

The expressions below for IX , IY , IZ , $WXMIN$, $WYMIN$, $WZMIN$, $SHARY$ and $SHARZ$ are taken from Ref. /1/. The expressions for $SHCENY$ and $SHCENZ$ are taken from Ref. /2/.

B 1.4 Double-bottom section

B 1.4.1 Sectional Dimensions

HZ	Height
------	--------

TY	Thickness of web
TB	Thickness of bottom plate
TT	Thickness of top plate
BY	Effective width of plates
SFY	Shear factor y-direction
SFZ	Shear factor z-direction

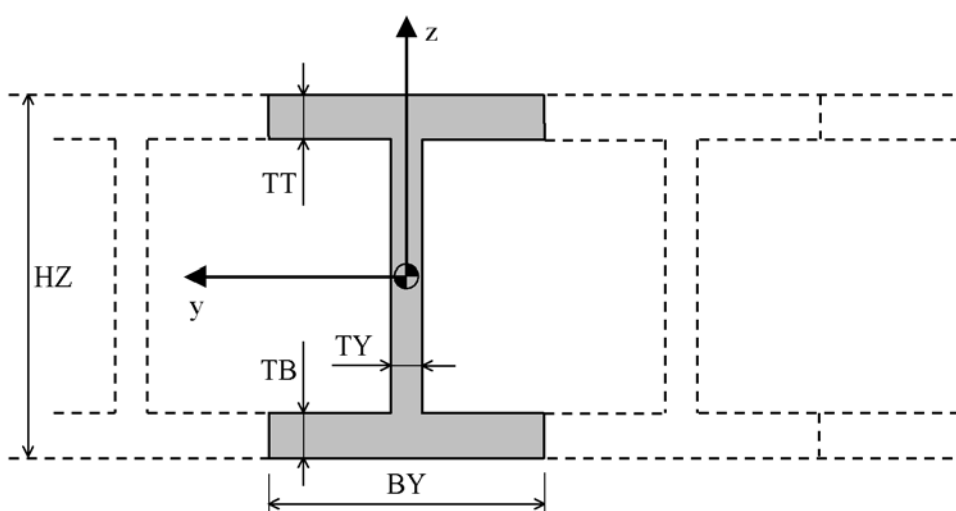


Figure B.4 Double-bottom section

B 1.4.2 Sectional Parameters Computed

The calculation procedure for the double bottom section is the same as for the I section for computation of all parameters except IX and $WXMIN$. In the formulae below IXI is the IX for the I section.

B 1.5 I (or H) section

B 1.5.1 Sectional Dimensions

HZ	Height
BT	Width of top flange
TT	Thickness of top flange
TY	Thickness of web
BB	Width of bottom flange

TB	Thickness of bottom flange
SFY	Shear factor y-direction
SFZ	Shear factor z-direction

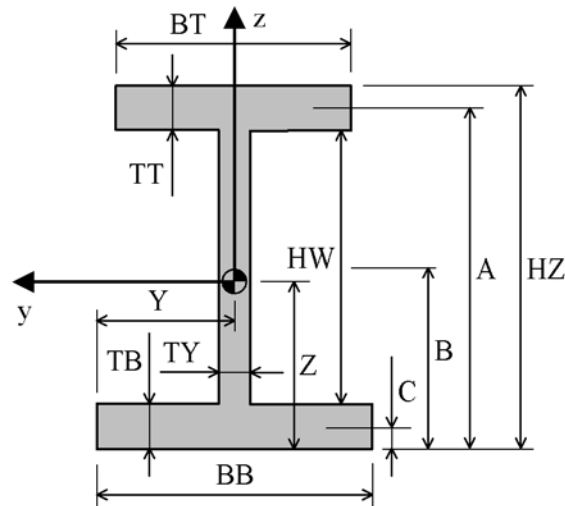


Figure B.5 I (or H) section

B 1.5.2 Sectional Parameters Computed

The expressions below for IX , IY , IZ , $WXMIN$, $WYMIN$, $WZMIN$, $SHARY$, $SHARZ$ and $SHCENY$ are taken from Ref. /1/. The expression for $SHCENZ$ is taken from Ref. /2/.

B 1.6 L section

B 1.6.1 Sectional Dimensions

HZ	Height
TY	Thickness of web
BY	Width of flange
TZ	Thickness of flange
SFY	Shear factor y-direction
SFZ	Shear factor z-direction
$POSWEB$	=1 for web location in positive y-direction, otherwise =-1

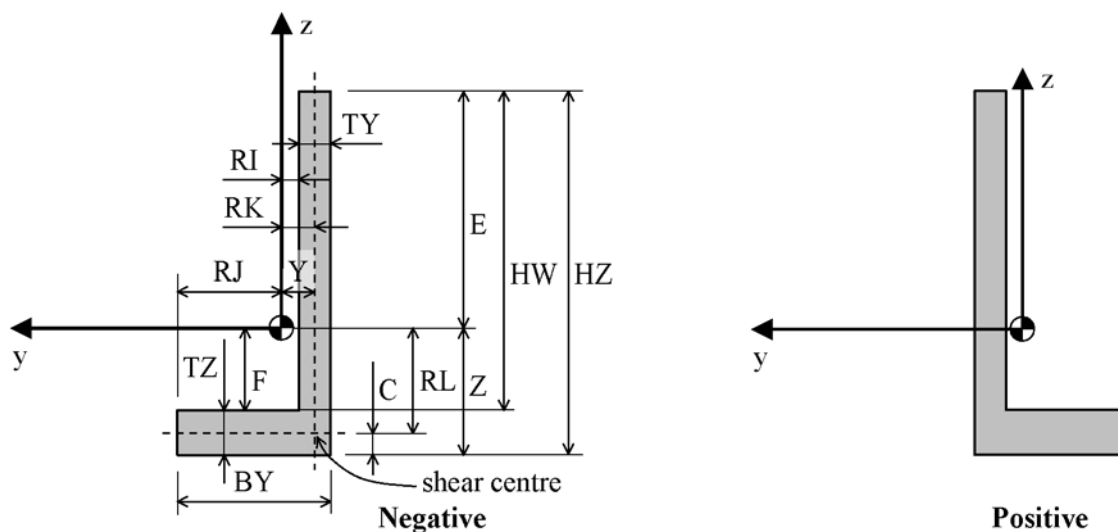


Figure B.6 L section

B 1.6.2 Sectional Parameters Computed

The expressions below for I_X , I_Y , I_Z , W_{XMIN} , W_{YMIN} , W_{ZMIN} , $SHARY$, $SHARZ$ and $SHCENY$ are taken from Ref. /1/. The expression for $SHCENZ$ is taken from Ref. /2/.

B 1.7 Pipe section

B 1.7.1 Sectional Dimensions

DY	Outer diameter
T	Thickness of wall
SFY	Shear factor y-direction
SFZ	Shear factor z-direction

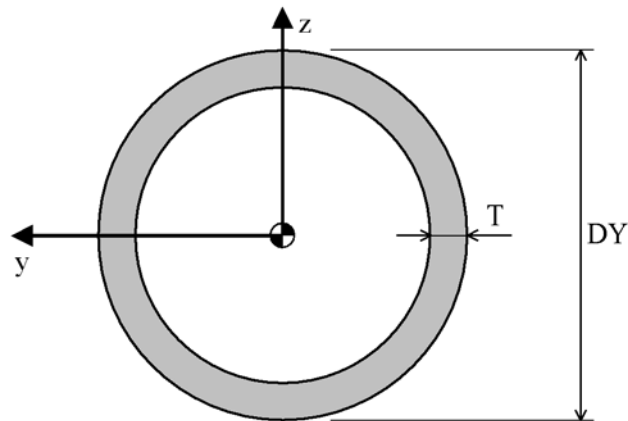


Figure B.7 Pipe section

B 1.7.2 Sectional Parameters Computed

The expressions below for I_X , I_Y , I_Z , W_{XMIN} , W_{YMIN} , W_{ZMIN} , $SHARY$, $SHARZ$, $SHCENY$ and $SHCENZ$ are taken from Ref. /1/.

B 1.8 Un-symmetrical I section

B 1.8.1 Sectional Dimensions

HZ	Height
BT	Width of top flange
BTA (B1 in Figure 5.29)	Width of part of top flange along positive y-axis
TT	Thickness of top flange
TY	Thickness of web
BB	Width of bottom flange
BBA (B2 in Figure 5.29)	Width of part of bottom flange along positive y-axis
TB	Thickness of bottom flange
SFY	Shear factor y-direction
SFZ	Shear factor z-direction

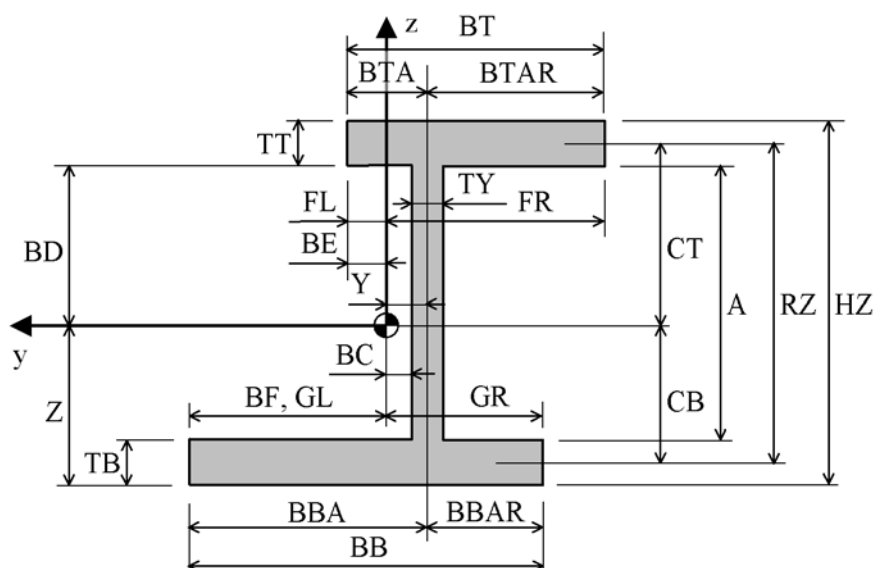


Figure B.8 Un-symmetrical I section

B 1.8.2 Sectional Parameters Computed

The expressions below for I_X , I_Y , I_Z , W_{XMIN} , W_{YMIN} , W_{ZMIN} , $SHARY$ and $SHARZ$ are taken from Ref. /1/. The expressions for $SHCENY$ and $SHCENZ$ are taken from Ref. /3/.

B 2 Units

A SESAM analysis is based on a set of consistent units. The units to use must be determined before commencing the modelling and these units must be adhered to throughout the analysis project, i.e. in all SESAM programs employed. The basis for determining a set of consistent units and some examples are given below.

The fundamental equation is:

$$\text{FORCE} = \text{MASS} \cdot \text{ACCELERATION}$$

In terms of the fundamental units of mass (M), length (L) and time (T) this equation may be written:

$$F = M \cdot L / T^2$$

Force, stress, density, etc. are not fundamental units and must be derived in terms of the fundamental units of M, L and T.

The first step in determining a set of consistent units is to select fundamental units. Input values to the programs such as steel density and Young's modulus (input to Preframe) and water density and gravity (input to Wajac) must then be determined in terms of these fundamental units.

Whenever possible it is simplest to use the SI units or multiples of the SI units:

- length in metres (m)

- mass in kilograms (kg)
- time in seconds (s)

A force will then be in Newton (N):

$$1 \text{ N} = 1 \text{ kg m} / \text{s}^2$$

B 2.1 Example

A model has been generated with centimetres (cm) as length unit. We want our output force unit to be tonnes-force (tonnef) and thus need to know which values for Young's modulus and steel density to specify in Preframe and which values for gravity and water density to specify in Wajac.

We first determine what our fundamental units of M, L and T are:

L is in centimetres (cm)

T is chosen to be in seconds (s)

We have already chosen the force unit to be tonnes-force (tonnef) and we know that:

$$1 \text{ tonnef} = 9810 \text{ kg m} / \text{s}^2 \approx 10000 \text{ kg m} / \text{s}^2$$

Insert this in the fundamental equation $F = M \cdot L / T^2$:

$$10000 \text{ kg m} / \text{s}^2 = M \cdot \text{cm} / \text{s}^2$$

$$10000 \text{ kg} \cdot 100 \text{ cm} / \text{s}^2 = M \cdot \text{cm} / \text{s}^2$$

Hence:

$$M = 10000 \cdot 100 \text{ kg} = 10^6 \text{ kg}$$

So our fundamental units are:

M in 10^6 kg

L in cm

T in s

The next step is to determine the density, Young's modulus, etc. in terms of our fundamental units.

Steel density, ρ :

Density = Mass / Volume = M / L^3 , thus the derived density shall be in $10^6 \text{ kg} / \text{cm}^3$

$$\rho_{\text{steel}} = 7850 \text{ kg} / \text{m}^3 = 7.85 \cdot 10^{-9} (10^6 \text{ kg} / \text{cm}^3)$$

Young's modulus, E:

Young's Modulus = Force / Area = $(M \cdot L / T^2) / L^2 = M / (L \cdot T^2)$, thus the derived Young's modulus shall be in $10^6 \text{ kg} / (\text{cm} \cdot \text{s}^2)$.

$$E = 2.1 \cdot 10^{11} \text{ N} / \text{m}^2 = 2.1 \cdot 10^{11} \text{ kg m} / (\text{s}^2 \cdot \text{m}^2) = 2.1 \cdot 10^{11} \text{ kg} / (\text{m} \cdot \text{s}^2)$$

Then in our derived units: $E = 2.1 \cdot 10^{11} / 10^6 \cdot 100 (10^6 \text{ kg} / \text{cm} \cdot \text{s}^2) = 2.1 \cdot 10^3 (10^6 \text{ kg} / \text{cm} \cdot \text{s}^2)$

Gravity:

Gravity = Acceleration = L / T^2 , thus our derived gravity unit shall be cm / s^2 .

$$g = 9.81 \text{ m} / \text{s}^2 = 9.81 \cdot 10^2 (\text{cm} / \text{s}^2) = 981 (\text{cm} / \text{s}^2)$$

Sea water density:

Density = Mass / Volume = M / L^3 , thus the derived density shall be in $10^6 \text{ kg} / \text{cm}^3$.

$$\rho_{\text{water}} = 1025 \text{ kg} / \text{m}^3 = 1.025 \cdot 10^{-9} (10^6 \text{ kg} / \text{cm}^3)$$

B 2.2 Consistent Sets of Units

Tables over sets of consistent units are provided below.

Nomenclature:

cm	centimetres
E	Young's modulus
kg	kilograms
kgf	kilograms-force
L	fundamental length symbol
m	metres
mm	millimetres
M	fundamental mass symbol
N	Newtons
s	seconds
t	tonnes
tonnef	tonnes-force
T	fundamental time symbol

rho

density

Table B.1 Examples of consistent units, time unit is second

Length unit L	Mass unit M	Force unit $M L / T^2$	Typical program input values	
			Density of steel (Mass/Volume) M / L^3	Young's modulus for steel (Force/Area) $M / (L \cdot T^2)$
m	kg	1 N	$7.85 \cdot 10^3$	$2.10 \cdot 10^{11}$
m	$10^3 \text{ kg} = 1 \text{ t}$	10^3 N	7.85	$2.10 \cdot 10^8$
cm	kg	10^{-2} N	$7.85 \cdot 10^{-3}$	$2.10 \cdot 10^9$
cm	$10^3 \text{ kg} = 1 \text{ t}$	$10 \text{ N} \approx 1 \text{ kgf}$	$7.85 \cdot 10^{-6}$	$2.10 \cdot 10^6$
mm	kg	10^{-3} N	$7.85 \cdot 10^{-6}$	$2.10 \cdot 10^8$
mm	$10^3 \text{ kg} = 1 \text{ t}$	1 N	$7.85 \cdot 10^{-9}$	$2.10 \cdot 10^5$
cm	10^2 kg	1 N	$7.85 \cdot 10^{-5}$	$2.10 \cdot 10^7$
m	10^4 kg	1 tonnef $\approx 10000 \text{ N}$	$7.85 \cdot 10^{-1}$	$2.10 \cdot 10^7$
cm	10^6 kg	1 tonnef $\approx 10000 \text{ N}$	$7.85 \cdot 10^{-9}$	$2.10 \cdot 10^3$
mm	10^7 kg	1 tonnef $\approx 10000 \text{ N}$	$7.85 \cdot 10^{-13}$	2.10
m	10 kg	1 kgf $\approx 10 \text{ N}$	$7.85 \cdot 10^2$	$2.10 \cdot 10^{10}$
cm	10^3 kg	1 kgf $\approx 10 \text{ N}$	$7.85 \cdot 10^{-6}$	$2.10 \cdot 10^6$
mm	10^4 kg	1 kgf $\approx 10 \text{ N}$	$7.85 \cdot 10^{-10}$	$2.10 \cdot 10^4$

REFERENCES

- 1 W. Beitz, K.H. Küttner:
"Dubbel, Taschenbuch für den Maschinenbau"
17. Auflage (17th ed.)
Springer-Verlag 1990
- 2 Arne Selberg:
"Stålkonstruksjoner"
Tapir 1972
- 3 S. Timoshenko:
"Strength of Materials, Part I, Elementary Theory and Problems"
Third Edition 1995
D. Van Nostrand Company Inc.